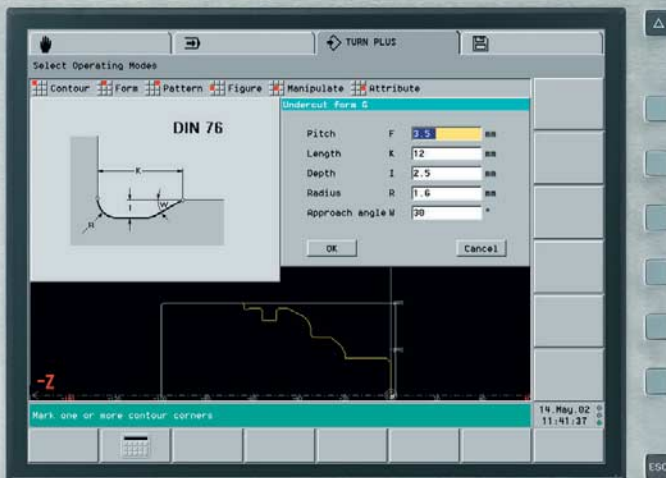




HEIDENHAIN



HEIDENHAIN



CNC PILOT 4290

NC Software
368 650-xx
V7

User's Manual

English (en)
10/2004

Data input keypad



Manual operating mode



Automatic mode



Programming modes (DIN PLUS, simulation, TURN PLUS)



Organization modes (parameter, service, transfer)



Display error status



Call the info system



ESC (escape)

- Go back by one menu level
- Close dialog box, do not save



INS (insert)

- Insert list element
- Close dialog box, save data



ALT (alter)

- Change the list element



DEL (delete)

- Deletes the list element
- deletes the selected character or the character to the left of the cursor.



... 9 Numbers for value input and soft-key selection



Decimal point



Minus as algebraic sign



"Continue key" for special functions (e.g. marking)



Arrow keys



Page forward, page backward

- Change to previous/next screen page
- Change to previous/next dialog box
- Switch between input windows



Enter – Confirmation of input

Machine operating panel



Cycle Start



Cycle Stop



Feed Stop



Spindle Stop



Spindle on – M3/M4 direction



Spindle jog – M3/M4 direction (The spindle turns until you press the key.)



Manual direction keys +X/-X



Manual direction keys +Z/-Z



Manual direction keys +Y/-Y



Rapid traverse key



Slide change key



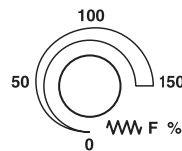
Spindle change key



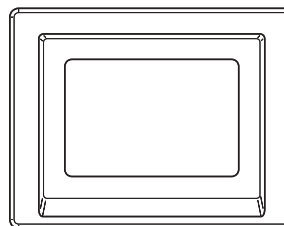
Spindle speed at the programmed value



Increase/decrease spindle speed by 5%



Override dial for feed rate



Touch pad with right and left mouse key



HEIDENHAIN



CNC PILOT 4290, Software and Functions

This manual describes functions that are available in the CNC PILOT 4290 with NC software number 368 650-xx (Release 7.0). For programming the Y-axis, please refer to the User's Manual "CNC PILOT 4290 with Y-Axis". It is not described in this manual.

The machine manufacturer adapts the features offered by the control to the capabilities of the specific lathe by setting machine parameters. Therefore, some of the functions described in this manual may not be among the features provided by the CNC PILOT on your machine tool.

Some of the CNC PILOT functions which are not available on every machine are:

- Machining with the C axis
- Machining with the Y-axis
- Full-surface machining
- Tool monitoring
- Graphically supported interactive contour definition
- Automatic or graphically supported interactive DIN PLUS program generation

Please contact your machine manufacturer for detailed information on the features that are supported by your machine tool.

Many machine manufacturers and HEIDENHAIN offer programming courses for the CNC PILOT controls. We recommend these courses as an effective way of improving your programming skills and sharing information and ideas with other CNC PILOT users.

HEIDENHAIN also offers the PC software DataPilot 4290, which is designed for use with the CNC PILOT 4290. The DataPilot is suitable for both shop-floor programming as well as off-location program creation and testing. It is also ideal for training purposes. DataPilot can be run on WINDOWS 95, WINDOWS 98, WINDOWS ME, WINDOWS NT 4.0 or WINDOWS 2000.

Intended place of operation

The CNC PILOT 4290 complies with EN 55022, Class A, and is intended primarily for operation in industrially zoned areas.

Contents

Introduction and Fundamentals	1
Basics of operation	2
Manual Control and Automatic Modes	3
DIN PLUS	4
Graphic Simulation	5
TURN PLUS	6
Parameter	7
Operating Resources	8
Service and Diagnosis	9
Transfer	10
Tables and overviews	11

1 Introduction and Fundamentals..... 1

- 1.1 The CNC PILOT..... 2
- 1.2 The Operating Modes..... 5
- 1.3 Expansion Stages (Options)..... 6
- 1.4 Fundamentals..... 7
- 1.5 Tool Dimensions..... 10

2 Basics of Operation..... 11

- 2.1 User Interface..... 12
 - 2.1.1 Screen Displays..... 12
 - 2.1.2 Controls and Displays..... 13
 - 2.1.3 Selection of Operating Modes..... 14
 - 2.1.4 Selection of Functions, Data Input..... 14
- 2.2 The Info System..... 16
- 2.3 The Error System..... 17
 - 2.3.1 Direct Error Messages..... 17
 - 2.3.2 Error Display, PLC Display..... 17
- 2.4 Data Backup..... 19
- 2.5 Explanation of Terms..... 19

3 Manual Control and Automatic mode..... 21

- 3.1 Switch-On, Switch-Off, Reference Run..... 22
 - 3.1.1 Switch-On and Traversing the Reference Marks..... 22
 - 3.1.2 Switch-Off..... 23
- 3.2 Manual Operating Mode..... 24
 - 3.2.1 Entering machine data..... 25
 - 3.2.2 M Commands..... 25
 - 3.2.3 Manual Turning Operations..... 26
 - 3.2.4 Handwheel..... 26
 - 3.2.5 Spindle and Axis Direction Keys..... 27
 - 3.2.6 Slide/Spindle change key..... 27
- 3.3 Tool Lists, Tool Life Management..... 28
 - 3.3.1 Setting Up a Tool List..... 29
 - 3.3.2 Comparing a Tool List with an NC Program..... 31
 - 3.3.3 Transferring the Tool List from an NC Program..... 32
 - 3.3.4 Tool Life Management..... 33
- 3.4 Setup Functions..... 34
 - 3.4.1 Defining the Tool Change Position..... 34
 - 3.4.2 Shifting the Workpiece Datum..... 35
 - 3.4.3 Defining the protection zone..... 36
 - 3.4.4 Setting up the Chucking Table..... 37
 - 3.4.5 Setting up Machine Dimensions..... 38
 - 3.4.6 Measuring Tools..... 39

3.5	Automatic Mode of Operation.....	41
3.5.1	Program Selection.....	41
3.5.2	Defining a Start Block.....	42
3.5.3	Program Sequence Modification.....	43
3.5.4	Compensation.....	44
3.5.5	Tool Life Management.....	45
3.5.6	Inspection Mode.....	46
3.5.7	Block Display.....	48
3.5.8	Graphic Display.....	49
3.5.9	Post-Process Measuring Status Display.....	51
3.6	Machine Display.....	52
3.7	Load Monitoring.....	54
3.7.1	Reference Machining.....	54
3.7.2	Production Using Load Monitoring.....	55
3.7.3	Editing Limit Values.....	56
3.7.4	Analyzing Reference Machining.....	57
3.7.5	Machining Using Load Monitoring.....	57
3.7.6	Load Monitoring Parameters.....	58

4 DIN PLUS..... 59

4.1	DIN Programming.....	60
4.1.1	Introduction.....	60
4.1.2	DIN PLUS Screen.....	61
4.1.3	Linear and Rotary Axes.....	62
4.1.4	Units of Measurement.....	63
4.1.5	Elements of the DIN Program.....	63
4.2	Programming Notes.....	65
4.2.1	Parallel Editing.....	65
4.2.2	Address Parameters.....	65
4.2.3	Contour Programming.....	66
4.2.4	Tool Programming.....	68
4.2.5	Fixed cycles.....	69
4.2.6	NC Subprograms.....	70
4.2.7	Template Control.....	70
4.2.8	NC Program Interpretation.....	70
4.3	The DIN PLUS Editor.....	71
4.3.1	Main Menu.....	72
4.3.2	Geometry Menu.....	75
4.3.3	Machining Menu.....	76
4.3.4	Block Menu.....	77

4.4	Program Section Codes.....	79
4.4.1	PROGRAMMKOPF [PROGRAM HEAD].....	79
4.4.2	TURRET.....	80
4.4.3	CHUCKING EQUIPMENT.....	82
4.4.4	Contour Definition.....	82
4.4.5	BEARBEITUNG [MACHINING].....	83
4.4.6	UNTERPROGRAMM [SUBPROGRAM].....	83
4.5	Geometry Commands.....	84
4.5.1	Definition of Blank.....	84
4.5.2	Basic Contour Elements.....	84
4.5.3	Contour Form Elements.....	86
4.5.4	Help Commands for Contour Definition.....	92
4.5.5	Contour Position.....	95
4.5.6	Front and Rear Face Contours.....	96
4.5.7	Lateral Surface Contours.....	102
4.5.8	Circular Pattern with Circular Slots.....	108
4.6	Machining Commands.....	110
4.6.1	Assigning the Contour to the Operation.....	110
4.6.2	Tool Positioning without Machining.....	110
4.6.3	Simple Linear and Circular Movements.....	111
4.6.4	Feed Rate and Spindle Speed.....	113
4.6.5	Cutter Radius Compensation (TRC/MCRC).....	115
4.6.6	Zero Point Shift.....	116
4.6.7	Oversizes, Safety Clearances.....	118
4.6.8	Tools, Types of Compensation.....	120
4.7	Turning Cycles.....	122
4.7.1	Contour-Based Turning Cycles.....	122
4.7.2	Simple Turning Cycles.....	134
4.8	Thread Cycles.....	140
4.9	Drilling cycles.....	143
4.10	C-Axis Machining.....	148
4.10.1	General C-Axis Functions.....	148
4.10.2	Front/Rear Face Machining.....	149
4.10.3	Lateral Surface Machining.....	150
4.11	Milling Cycle Group.....	152
4.12	Special functions.....	159
4.12.1	Chucking Equipment in Simulation.....	159
4.12.2	Slide Synchronization.....	160
4.12.3	Spindle Synchronization, Workpiece Transfer.....	161
4.12.4	Contour Follow-Up.....	164
4.12.5	In-Process Measuring.....	165

4.12.6 Post-Process Measuring.....	166
4.12.7 Load Monitoring.....	167
4.13 Other G Functions.....	168
4.14 Data Input and Data Output.....	173
4.14.1 Input/Output of # Variables.....	173
4.14.2 Input/Output of V Variables.....	174
4.15 Programming Variables.....	175
4.15.1 # Variables.....	175
4.15.2 V Variables.....	177
4.15.3 Program Branches, Program Repeats, Conditional Block Execution.....	179
4.16 Subprograms.....	182
4.17 M Functions.....	183
4.18 Programming Notes and Examples.....	184
4.18.1 Programming Machining Cycles.....	184
4.18.2 Contour Repetitions.....	184
4.18.3 Full-Surface Machining.....	187

5 Graphic Simulation..... 195

5.1 Simulation Mode of Operation.....	196
5.1.1 Graphic Elements, Displays.....	197
5.1.2 Basics of Operation.....	200
5.2 Main Menu.....	201
5.3 Contour Simulation.....	203
5.3.1 Contour-Simulation Functions.....	203
5.3.2 Dimensioning.....	204
5.4 Machining Simulation.....	205
5.5 Motion Simulation.....	207
5.6 Zoom Function.....	208
5.7 3-D View.....	209
5.8 Checking the	
5.9 Time Calculation.....	212
5.10 Synchronous Point Analysis.....	213

6 TURN PLUS..... 215

6.1 TURN PLUS Mode of Operation.....	216
6.2 Program Management.....	217
6.2.1 TURN PLUS Files.....	217
6.2.2 Program Head.....	218
6.3 Workpiece Description.....	219
6.3.1 Entering the Contour of a Blank Part.....	219
6.3.2 Input of the Finished Part Contour.....	220
6.3.3 Superimposing form elements.....	221
6.3.4 Integrating a Contour Train.....	222

6.3.5	Entering Contours Machined with the C Axis.....	223
6.3.6	Basics of Operation.....	225
6.3.7	Help Functions for Element Definition.....	226
6.4	Contours of Workpiece Blanks.....	228
6.5	Contour of Finished Part.....	229
6.5.1	Basic Contour Elements.....	229
6.5.2	Form elements.....	232
6.5.3	Overlay Elements.....	239
6.6	C-Axis Contours.....	242
6.6.1	Contours on the Front and Rear Face.....	242
6.6.2	Contours of the Lateral Surface.....	249
6.7	Manipulating Contours.....	256
6.7.1	Editing the Contours of a Blank Part.....	256
6.7.2	Trimming.....	256
6.7.3	Change.....	258
6.7.4	Deleting.....	259
6.7.5	Inserting.....	260
6.7.6	Transformations.....	261
6.7.7	Connect.....	262
6.7.8	Resolve.....	262
6.8	Importing DXF Contours.....	263
6.8.1	Fundamentals.....	263
6.8.2	Configuring the DXF Import.....	264
6.8.3	DXF-Import.....	266
6.8.4	Transferring and Organizing DXF Files.....	266
6.9	Assigning attributes.....	267
6.9.1	Attributes for Workpiece Blanks.....	267
6.9.2	Oversize.....	267
6.9.3	Feed rate/peak-to-valley height.....	267
6.9.4	Precision stop.....	268
6.9.5	Separation Points.....	268
6.9.6	Machining Attributes.....	269
6.10	User Aids.....	273
6.10.1	Calculator.....	273
6.10.2	Digitizing.....	274
6.10.3	Inspector – Checking Contour Elements.....	274
6.10.4	Unresolved Contour Elements.....	275
6.10.5	Error Messages.....	276
6.11	Preparing a Machining Process.....	277
6.11.1	Chuckling a Workpiece.....	277
6.11.2	Setting Up a Tool List.....	284

6.12 Interactive Working Plan Generation (IWG).....	286
6.12.1 Tool call.....	287
6.12.2 Cutting Data.....	288
6.12.3 Cycle specification.....	288
6.12.4 Roughing.....	289
6.12.5 Recessing.....	294
6.12.6 Drilling.....	299
6.12.7 Finishing.....	301
6.12.8 Thread Machining(G31).....	306
6.12.9 Milling.....	307
6.12.10 Special Machining Tasks (SM).....	309
6.13 Automatic Working Plan Generation (AWG).....	310
6.13.1 Generating a Machining Plan.....	310
6.13.2 Machining Sequence.....	311
6.14 Control Graphics.....	321
6.15 Configuration.....	322
6.16 Machining Information.....	324
6.16.1 Tool Selection, Turret Assignment.....	324
6.16.2 Cutting Parameters.....	325
6.16.3 Coolant.....	325
6.16.4 Hollowing.....	326
6.16.5 Inside Contours.....	326
6.16.6 Drilling.....	328
6.16.7 Full-Surface Machining.....	328
6.16.9 Shaft Machining.....	330
6.17 Example.....	332

7 Parameters..... 337

7.1 Parameter Mode of Operation.....	338
7.1.1 Parameters.....	338
7.1.2 Editing Parameters.....	339
7.2 Machine Parameters.....	341
7.3 Control Parameters.....	348
7.4 Set-Up Parameters.....	355
7.5 Machining Parameters.....	357

8 Operating Resources..... 371

- 8.1 Tool Database..... 372
 - 8.1.1 Tool Editor..... 372
 - 8.1.2 Tool Types (Overview)..... 375
 - 8.1.3 Tool Parameters..... 377
 - 8.1.4 Multipoint Tools, Tool Life Monitoring..... 384
 - 8.1.5 Explanation of Tool Data..... 385
 - 8.1.6 Tool Holder, Mounting Position..... 387
- 8.2 Chucking Equipment Database..... 390
 - 8.2.1 Chucking Equipment Editor..... 390
 - 8.2.2 Chucking Equipment Data..... 392
- 8.3 Technology Database (Cutting Values)..... 399

9 Service and Diagnosis..... 401

- 9.1 Service Mode of Operation..... 402
- 9.2 Service Functions..... 402
 - 9.2.1 Access Authorization..... 402
 - 9.2.2 System Service..... 403
 - 9.2.3 Fixed-Word Lists..... 404
- 9.3 Maintenance System..... 405
- 9.4 Diagnosis..... 408

10 Transfer..... 411

- 10.1 The Transfer Mode of Operation..... 412
- 10.2 Transfer Systems..... 413
 - 10.2.1 General Information..... 413
 - 10.2.2 Configuring for Data Transfer..... 414
- 10.3 Data Transfer..... 417
 - 10.3.1 Enabling, Data Types..... 417
 - 10.3.2 Transmitting and Receiving Files..... 418
- 10.4 Parameters and Operating Resources..... 420
 - 10.4.1 Converting Parameters and Operating Resources..... 420
 - 10.4.2 Saving Parameters and Operating Resources..... 422
- 10.5 File Organization..... 423

11 Tables and overviews..... 425

- 11.1 Undercut and Thread Parameters..... 426
 - 11.1.1 Undercut DIN 76, Parameters..... 426
 - 11.1.2 Undercut DIN 509 E, Parameters..... 427
 - 11.1.3 Undercut DIN 509 F, Parameters..... 427
 - 11.1.4 Thread Parameters..... 428
 - 11.1.5 Thread Pitch..... 429
- 11.2 Technical Information..... 433
- 11.3 Peripheral Interfaces..... 437



1

Introduction and Fundamentals

1.1 The CNC PILOT

The CNC PILOT is a contouring control designed for lathes and turning centers. In addition to turning operations, you can perform milling and drilling operations with the C-axis or the Y-axis. The CNC PILOT supports parallel machining of up to 4 workpieces in programming, testing and production. Full-surface machining is supported on lathes with:

- Rotating gripper
- Movable opposing spindle
- Multiple spindles, slides and tool carriers

The CNC PILOT controls up to 6 slides, 4 spindles and 2 C axes.

Programming

Depending on the type and complexity of the parts to be machined and your organization, you can choose the type of programming best suited to your tasks.

In **TURN PLUS** you describe the contour of the blank and finished part with interactive graphics. Then you call the automatic working plan generation (AWG), and the NC program will be generated fully automatically at a keystroke. Alternately, you can choose the interactive working plan generation (IWG). When using the IWG, you determine the sequence of machining and other technical details.

Every working step is shown in the control graphics and can be corrected immediately. The result of program creation with TURN PLUS is a structured DIN PLUS program.

TURN PLUS minimizes the number of entries required, but it requires that the tool data and cutting data has already been entered.

If TURN PLUS fails to create the optimal NC program for technologically sophisticated machining operations, or if you primarily want to reduce the machining time, program the NC program in DIN PLUS.

DIN PLUS supports the separation of the geometric description from the machining of the workpiece. Powerful cycles are available for programming in DIN PLUS. The "simple geometry programming" function calculates coordinates if the dimensions used in the drawing are not suitable for NC programs.

Alternately, you can machine your workpiece in DIN PLUS with linear and circular movements and simple turning cycles, as you are accustomed to in conventional DIN programming.



Both TURN PLUS and DIN PLUS support machining with the C-axis or Y-axis and full-surface machining.

The **Graphic Simulation** feature enables you to subject your NC programs to a realistic test. The CNC PILOT displays the machining of up to 4 workpieces in the working space. Workpiece blanks and finished parts, chucking equipment and tools are shown to scale.

You can program your NC programs and test them -even during machining operations- directly on the machine.

Regardless of whether you are machining a simple or complex part, producing a single part or a series of parts, or a whole batch on a turning center, the CNC PILOT always gives you optimum support.

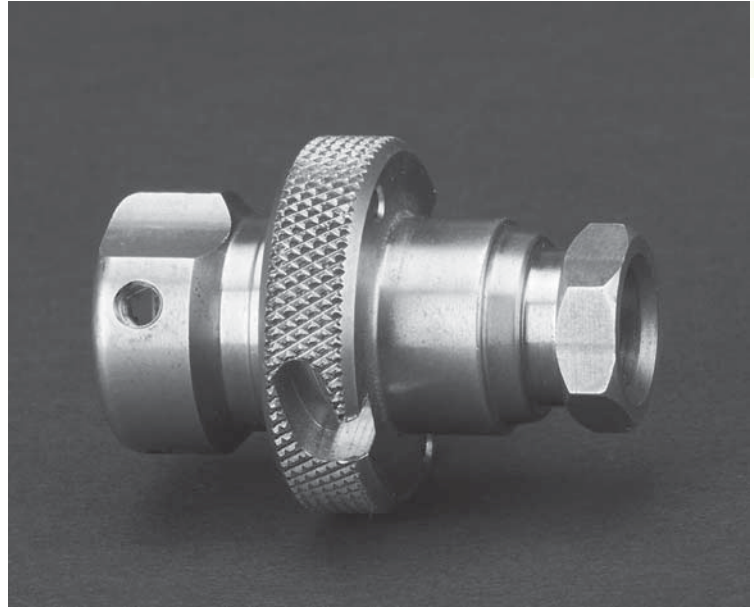
The C-axis

With a C-axis you can drill and mill a workpiece on its front, back and lateral surfaces.

During use of the C-axis, one axis interpolates linearly or circularly with the spindle in the given working plane, while the third axis interpolates linearly.

The CNC PILOT supports part program creation with the C-axis in:

- DIN PLUS
- TURN PLUS contour definition
- TURN PLUS working plan generation



The Y-axis

With a Y-axis you can drill and mill a workpiece on its front, back and lateral surfaces.

During use of the Y-axis, two axes interpolate linearly or circularly in the given working plane, while the third axis interpolates linearly. This enables you to machine slots or pockets, for example, with plane floors and perpendicular edges. By defining the spindle angle, you can determine the position of the milling contour on the workpiece.

The CNC PILOT supports part program creation with the Y-axis in:

- DIN PLUS
- TURN PLUS contour definition
- TURN PLUS working plan generation

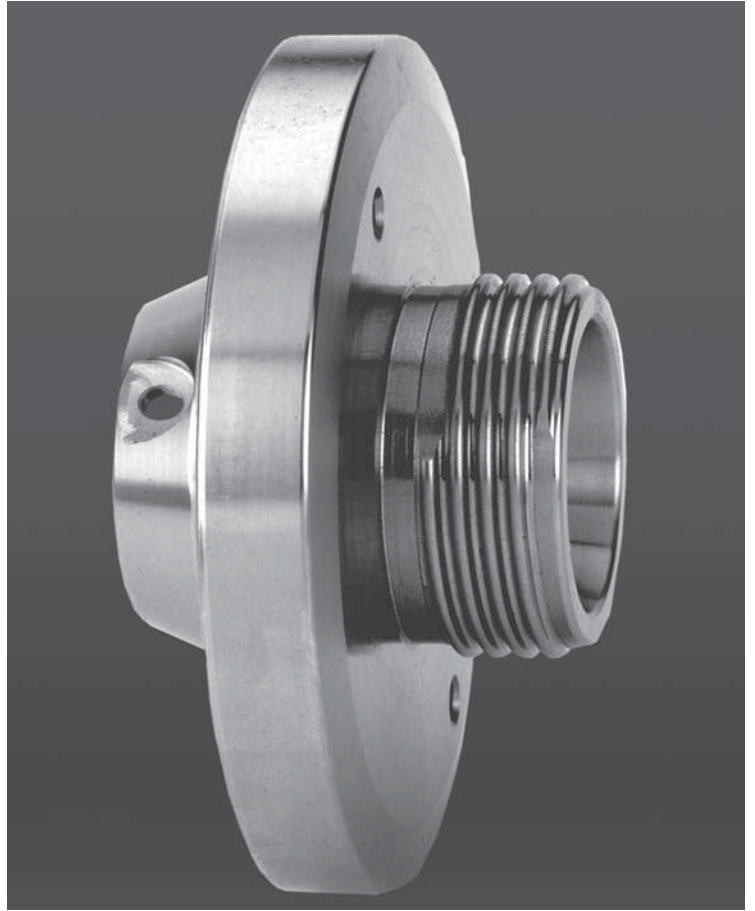


Full-surface machining

The CNC PILOT supports full-surface machining for all common machine designs. The features include angle-synchronous part transfer with rotating spindle, traversing to a stop, controlled parting, and coordinate transformation. This ensures efficient full-surface machining and simple programming.

The CNC PILOT supports full surface machining in:

- DIN PLUS
- TURN PLUS contour definition
- TURN PLUS working plan generation



1.2 The Operating Modes

The functions of the CNC PILOT are grouped into the following operating modes:



Manual operating mode

In the Manual mode you set up the machine and move the axes manually.



Automatic mode

The NC programs are run in Automatic mode. You control and monitor the machining of the workpiece.



DIN PLUS programming mode

In "DIN PLUS," you can create structured NC programs. You first define the geometry of the blank and finished part, and then program the individual operations.



Simulation programming mode

The Simulation mode shows a graphic representation of programmed contours, the paths of traverse and cutting operations. The working space, tools and chucking equipment are shown true to scale.

During simulation, the CNC PILOT calculates the **machining and idle-machine times** for every tool. For lathes with several slides, the **Synchronous point analysis** enables you to optimize your NC program.



TURN PLUS programming mode

In "TURN PLUS" you describe the contour of the workpiece using interactive graphics. For Automatic Working plan Generation (AWG), you select the material and chucking equipment. The CNC PILOT will generate the NC program automatically at a keystroke. As an alternative, you can create the working plan with the aid of interactive graphics (IAG).



Parameter organization mode

The system behavior of the CNC PILOT is controlled with parameters. In this mode, you set the parameters to adapt the control to your situation.

In addition, in this mode you describe the operating resources (tools and chucking equipment) and the cutting values.



Service organization mode

In "Service" mode, you log on for password-protected functions, select the conversational language and make the system settings. This operating mode also provides diagnostic functions for commissioning and checking the system.



Transfer organization mode

In "Transfer" you exchange the files with other systems, organize your programs and make data backups.

The actual control is not accessible to the machinist. You should know, however, that your CNC PILOT has an integrated hard disk on which all TURN PLUS and DIN PLUS programs that you enter are stored. This allows you to save a vast number of programs.

For data exchange and data backup, you can use the **Ethernet interface**. Data exchange is also possible over the **serial interface. (RS232)**.

1.3 Expansion Stages (Options)

The machine manufacturer configures the CNC PILOT according to the capabilities of the specific lathe. The following upgrades (options) are available, which enable you to adapt the control to your specific requirements:

■ TURN PLUS

Graphically supported interactive contour definition

- Graphic description of the workpiece for blank and finished part
- Geometry-programming function for calculating and displaying missing contour data
- Simple input of standard form elements like chamfers, rounding arcs, recesses, undercuts, threads, fits, etc.
- Easy-to-use transformations like shifting, rotating, mirroring or multiplying

DIN PLUS program generation with interactive graphics

- Selection of the appropriate machining method
- Selection of the tools and definition of the cutting data
- Direct graphic control of machining process
- Immediate compensation possibility

Automatic DIN PLUS program generation

- Automatic selection of tools
- Automatic generation of working plan

■ TURN PLUS – extension by C-axis or Y-axis

- C-axis: representation of programmed contour in the following views: XC plane (front/rear end) and ZC plane (unrolled surface)
- Y-axis: representation of programmed contour in the following views: XY plane (front/rear end) YZ plane (side view)
- Hole and figure patterns
- Fixed cycles
- Interactive or automatic working plan generation – also for machining with the C-axis or Y-axis

■ TURN PLUS – extension by opposing spindle

- Rechucking with expert program
- Interactive or automatic generation of working plan – also for rechucking and 2nd setup

■ In-process measuring

- With triggering probe
- For measuring tools
- For measuring workpieces

■ Post-process measuring

- Connection of measuring system via RS-232 interface
- Evaluation of measuring results in Automatic mode

Options can usually be retrofitted. Your machine manufacturer can give you more information on retrofitting.



This operating manual describes all options. The operating sequences described in this manual may therefore deviate from those on your machine whenever a certain option is not supported by your system.

1.4 Fundamentals

Axis designations

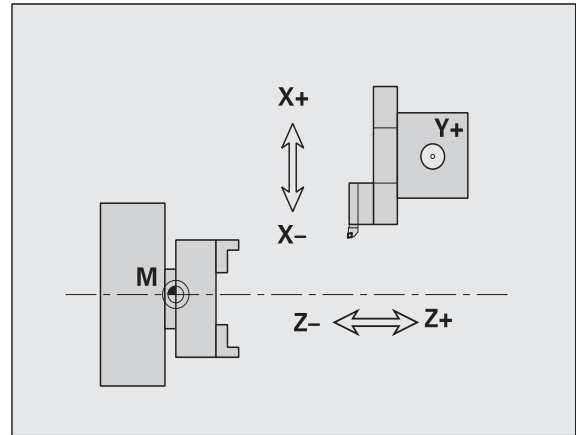
The cross slide is referred to as the **X-axis** and the saddle as the **Z-axis**.

All X-axis values that are displayed or entered are regarded as **diameters**. In TURN PLUS you can define whether the X-axis values are to be interpreted as diameters or radii.

Lathes with **Y-axis**: The Y-axis is perpendicular to the X-axis and Z-axis (Cartesian system).

When programming paths of traverse, remember to:

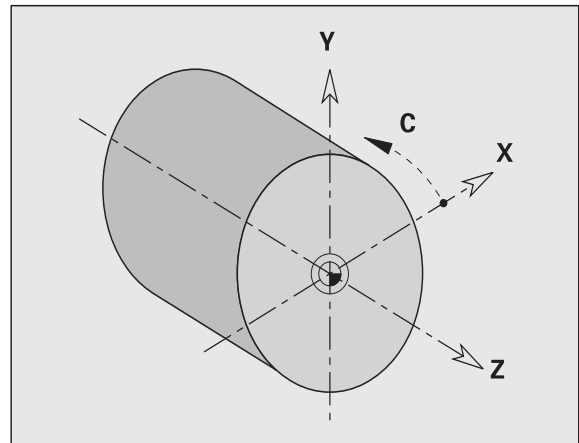
- Program a **positive value** to depart the workpiece.
- Program a **negative value** to approach the workpiece.



Coordinate system

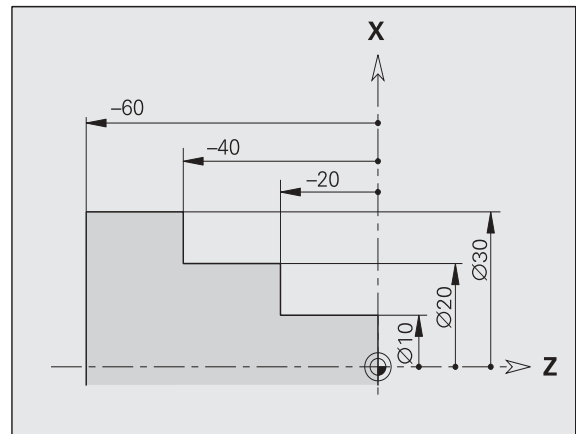
The coordinates entered for the **principal axes** X, Y and Z are referenced to the workpiece zero point – exceptions to this rule will be indicated.

Angles entered for the **C-axis** are referenced to the "zero point of the C-axis" (precondition: the C-axis has been configured as a principal axis).



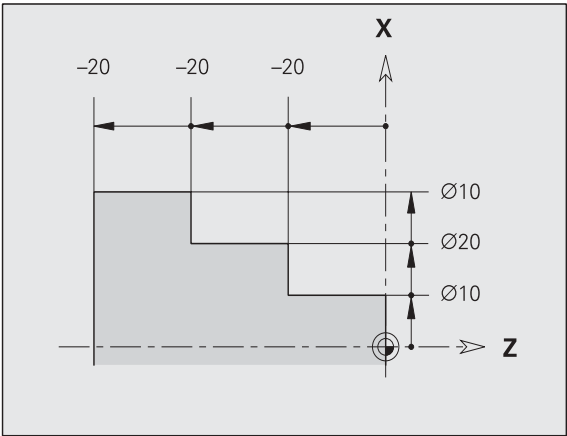
Absolute coordinates

If the coordinates of a position are referenced to the workpiece zero point, they are referred to as absolute coordinates. Each position on a workpiece is clearly defined by its absolute coordinates.



Incremental coordinates

Incremental coordinates are always referenced to the last programmed position. They specify the distance from the last active position and the subsequent position. Each position on a workpiece is clearly defined by its incremental coordinates.

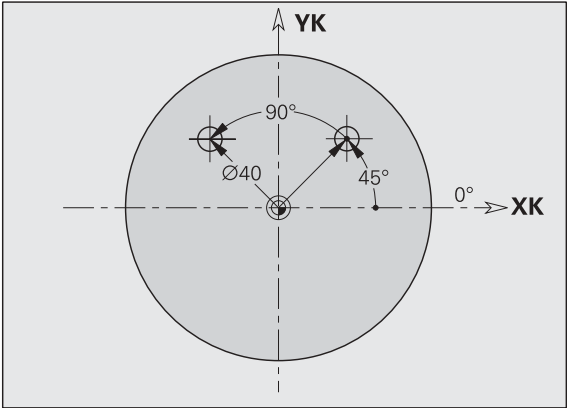


Polar coordinates

Positions located on the face or lateral surface can either be entered in Cartesian coordinates or polar coordinates.

When programming with polar coordinates, a position on the workpiece is clearly defined by the entries for diameter and angle.

You can enter polar coordinates as absolute or incremental values.



Units of measurement

You can program and operate the CNC PILOT either in the metric or inch system. The units of measurement listed in the table below apply to all inputs and displays.

Measure	Metric	inch
Coordinates	mm	inch
Lengths	mm	inch
Angles	Degrees	Degrees
Spindle speed	rpm	rpm
Cutting speed	m/min	ft/min
Feed per revolution	mm/rev	inch/rev
Feed per minute	mm/min	inch/min
Acceleration	m/s ²	ft/s ²

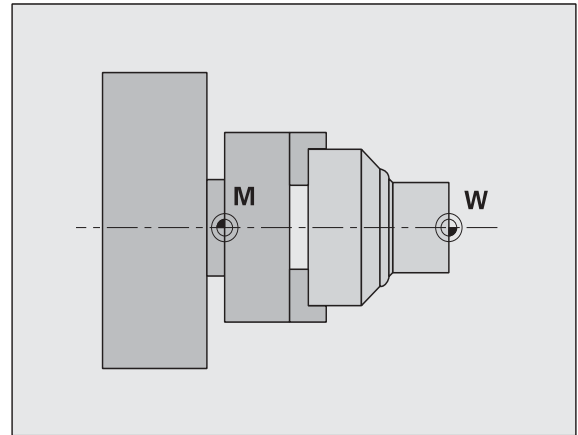
Machine reference points

Machine zero point

The point of intersection of the X-axis with the Z-axis is called the **machine zero point**. On a lathe, the machine zero point is usually the point of intersection of the spindle axis and the spindle surface. The machine zero point is designated with the letter M.

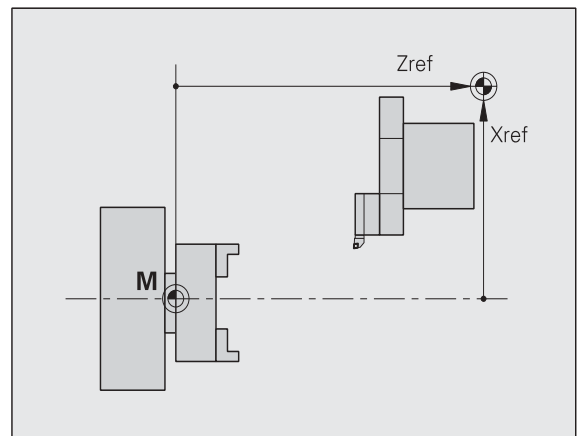
Workpiece zero point

For machining a workpiece, it is easier to reference all input data to a zero point located on the workpiece. By programming the zero point used in the workpiece drawing, you can take the dimensions directly from the drawing, without further calculation. This point is the "workpiece zero point." The workpiece zero point is designated with the letter W.



Reference marks

Whether the control "forgets" the positions of the machine axes when it is switched off depends on the position encoders used. If the positions are lost, you must pass over the fixed reference points after switching on the CNC PILOT. The system knows the distances of the reference points to the machine datum.



1.5 Tool Dimensions

The CNC PILOT requires information on the specific tools for a variety of tasks, such as calculating the cutting radius compensation or the proportioning of cuts.

Tool length

All position values that are programmed and displayed are referenced to the distance between the tool tip and workpiece zero point. Since the control only knows the absolute position of the tool carrier (slide), it needs the dimensions XE and ZE to calculate and display the position of the tool tip. For milling and drilling tools operating with the Y-axis, the CNC PILOT additionally needs the dimension in Y.

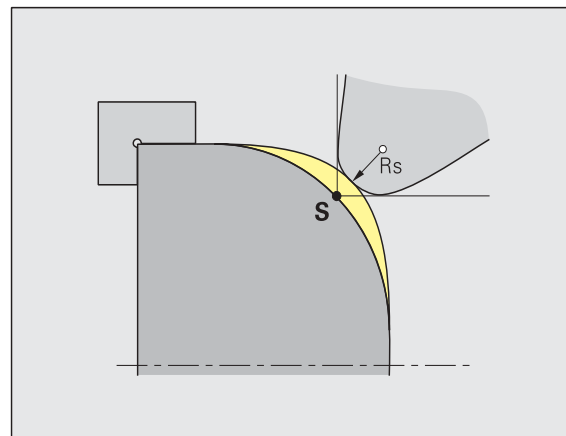
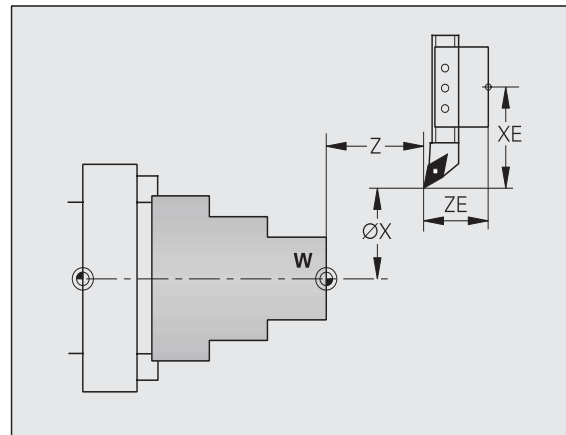
Tool compensation

The tool tip is subjected to wear during machining processes. To compensate for this wear, the CNC PILOT uses compensation values. The system automatically adds the compensation values to the values for length.

Tooth and cutter radius compensation (TRC)

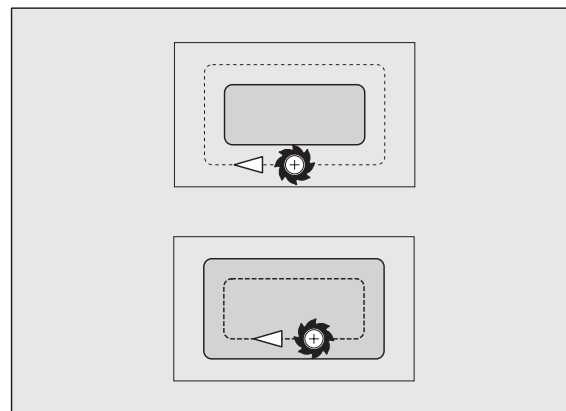
The tip of a lathe tool has a certain radius. When machining tapers, chamfers and radii, this results in inaccuracies which the CNC PILOT compensates with its cutting radius compensation function.

Programmed paths of traverse are referenced to the theoretical tool tip **S**. The TRC function compensates for this error by calculating a new path of traverse, the **equidistant line**.



Milling cutter radius compensation (MRC)

In milling operations, the outside diameter of the milling cutter determines the contour. When the MRC function is not active, the system defines the center of the cutter as reference point for the paths of traverse. The MRC function compensates for this error by calculating a new path of traverse, the **equidistant line**.





2

Basics of Operation

2.1 User Interface

2.1.1 Screen Displays

1 Operating mode line

Show the status of the operating modes.

- The active mode of operation is shown with a dark-gray background.
- Programming and organization modes:
 - The selected mode is shown at the right of the symbol
 - Additional information such as the selected program, submode, etc. are shown below the operating mode symbol.

2 Menu bar and pull-down menus

For function selection

3 Working window

Size and content depend on the operating mode. Some programming and organization modes overlap the machine display.

4 Machine display

Current status of the machine (tool position, the cycle and spindle situation, active tool, etc.). The machine display is configurable.

5 Status line

- Simulation, TURN PLUS: display of current settings or information on the next operating steps
- Other operating modes: display of the last error message

6 Calendar date and service "traffic light"

- Display of date and time
- A color background signals a error or a PLC message
- The "service traffic light" shows the servicing state of the machine (see "9.3 Maintenance system")

7 Soft-key row

Shows the current meaning of the soft keys.

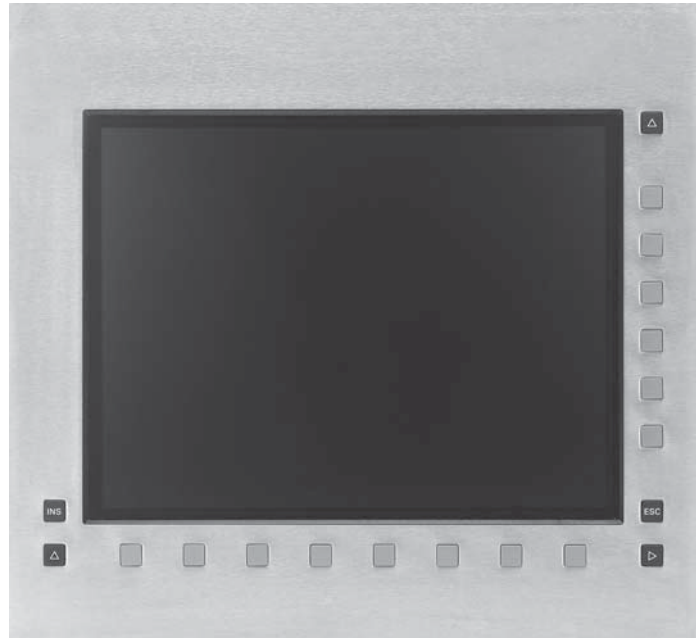
8 Vertical soft-key row

Shows the current meaning of the soft keys. For more information: see the machine manual



2.1.2 Controls and Displays

- **Screen** with
 - Horizontal and vertical **soft keys**: The meaning is shown above or next to the soft keys
 - Additional keys** (same function as on the operating panel):
 - ESC
 - INS
- **Operating panel** with
 - **Alphanumeric keyboard** with integrated **numeric keypad**
 - Keys for **Operating mode selection**
 - **Touch pad**: For cursor positioning (menu or soft key selection, selection from lists, selecting edit boxes, etc.)
- **Machine operating panel** with
 - Operating elements for the manual and automatic operation of the lathe (cycle keys, manual direction keys, etc.)
 - **Handwheel** for exact positioning in manual operation
 - **Override button** for feed-rate override



Operation of the touch pad

Normally, you can use the touch pad as an alternative to the cursor keys. In the following, the keys below the touch pad are referred to as the left and right mouse keys.

The functions and operation of the touch pad are similar to the mouse operation of the Windows operating systems.

- Single click of the left mouse key or single touch on the touch pad:
 - The cursor is positioned in lists or input windows.
 - Menu items, soft keys or buttons are activated
- Double-click of the left mouse key or double touch on the touch pad: In lists, the selected element is activated (the input window is activated)
- Single-click with the right mouse key:
 - Same function as the ESC key – prerequisite: the ESC key is allowed in this situation (for example to go back by one menu level)
 - Same function as the left mouse key when selection soft keys or buttons



2.1.3 Selection of Operating Modes

You can switch the operating mode at any time. After the change, the new mode starts in the function in which it was last exited.

In the **programming and organization modes** a difference is made between the following situations:

- No operating mode is selected (no entry next to the operating mode symbol): Select the desired mode from the menu.
- Operating mode selected (indicated next to the operating mode symbol): The functions of this operating mode are available. Within the programming or organization modes, you can switch the modes by soft key or by repeatedly pressing the corresponding mode key.

2.1.4 Selection of Functions, Data Input

Menu bar and pull-down menu

The individual menu items are preceded by a 9-field symbol with one field highlighted. This field represents the field on the numeric keypad. Press the key whose position corresponds to the position of the highlighted field.

The function selection begins in the menu row, then goes to the pull-down menus. In the pull-down menu, press again the numeric key assigned to the menu item – or alternatively, select the menu item with touch pad or with the “page up/page down” keys and press Enter.

Soft-key row





The meaning of the soft keys is dependent on the current operating situation.

Some soft keys work like “toggle switches”. A function is active when the associated field in the function-key row is highlighted in color. The setting remains in effect until the function is switched off.

List Operations

DIN PLUS programs, tool lists, parameter lists, etc. are displayed as lists. You can scroll through a list with the touch pad or arrow keys to check data, to select the position where you wish to enter data, or to highlight items for operations like deleting, copying, editing, etc.

After having selected the desired list position or a list item, press the ENTER, INS, ALT or DEL key to execute the operation.

Keys for operating mode selection:	
	Manual operating mode
	Automatic operating mode
	Programming modes
	Organization modes

Continued ►

Data Input

Data are entered and edited in **input windows**. An input window consists of several **input fields**. You position the cursor with the touch pad or with the page up/page down keys to the input box.

Once the cursor is located in the box, you can enter your data. Existing data are overwritten. With the right/left arrow keys you can place the cursor on a position **within** the input box in order to delete or add characters. The up/down arrow keys or Enter confirm and terminate the entry.

Some dialogs have more input fields than a window can show. In these cases, more than one input window appears on the screen, one superimposed on the other. You will recognize this through the window number in the top line. To toggle between input windows, use the Page Up/Page Dn keys.

By pressing the "OK" button, you confirm the data entered or edited. Independent of the position of the cursor, you can press the INS key as an alternative. If you leave the input window by pressing the "Cancel" button or the ESC key, entries or changes will be lost.

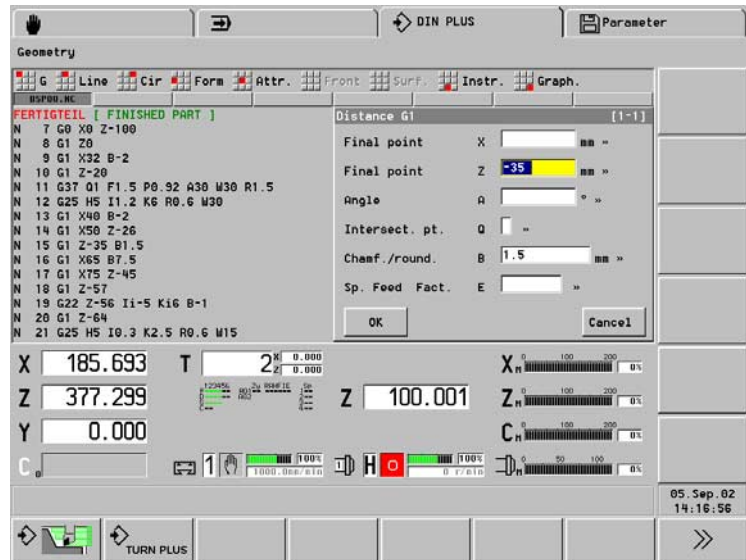
If the dialog consists of more than one input window, you already confirm the data when pressing the PageUp/PageDn key.

Buttons

The CNC PILOT allows you to choose various options via different buttons such as the "OK" and "Cancel" buttons for terminating a dialog box or the buttons contained in the "Extended inputs" window.

Select the required button and press ENTER.

Note: Instead of selecting the "OK" or "Cancel" button, you can press the INS or ESC key.



2.2 The Info System


The info system calls excerpts from the User's Manual to the screen. The system is structured in **info topics** comparable to the chapters of a book. In the top line of the information window, the topic you selected and the page number are shown.


The info system gives you information on the current operating situation (context-sensitive help). Also, you can select the info topics through the table of contents or the subject index. Simply select the desired topic or word and click "Topic select" (or Enter).

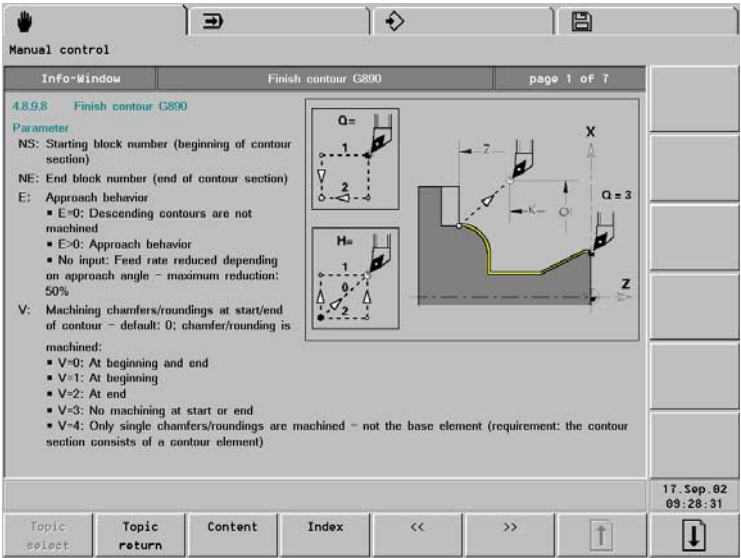
Cross references in the text are highlighted. Place the cursor on the desired cross reference and call the topic with "Topic select." "Topic return" switches back to the previous topic.

Error display



When an error message occurs, press the info key, or place the cursor on the error message in the "display of errors" and then press the info key, to get further information on the respective error.

 Call the info system

 End the info system



Soft keys

Topic select	<p>Calls the</p> <ul style="list-style-type: none"> Selected cross reference Topic from the table of contents Topic from the subject index
Topic return	Returns to the most recent info topic
Content	Calls the table of contents with the overview of info topics. The table of contents is arranged in several levels.
Index	Calls the subject index
<<	Switches to the previous topic.
>>	Switches to the next topic.
	(or page up key) previous info page
	(or page down key) next info page

2.3 The Error System

2.3.1 Direct Error Messages

Direct error messages appear whenever immediate error correction is possible. Confirm the message by pressing ENTER and correct the error. Example: The input value of the parameter is out of range.

Information of the error message:

- **Error description:** Explains the error
- **Error number:** For service inquiries
- **Time of day** When the error occurred (for your information).

Symbols

Warning

The program run/operation continues. The CNC PILOT indicates the problem.



Error

The program run/operation is stopped. You must correct the error before you can continue the current job.



2.3.2 Error Display, PLC Display

Error Display

If during the system start or during program run or other operation an error occurs, it is indicated in the date box, displayed in the status line, and saved in the error display.

The date and time remain highlighted in red until all of the errors have been canceled.

Notes on using TURN PLUS:



Opens the "error display"



Further information on the error marked with the cursor



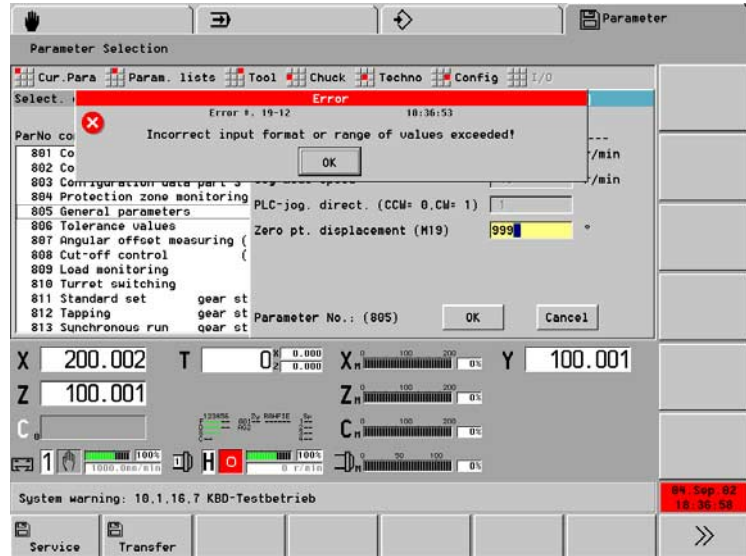
Exits the error display



Deletes the error message marked with the cursor



Deletes **all** error messages.



Continued ►

Information of the error message:

- **Error description:** Explains the error
- **Error number:** For service inquiries
- **Channel number:** Slide for which the error occurred.
- **Time of day** When the error occurred (for your information).
- **Error class** (only with errors):
 - **Background:** The message serves for information only, or it is a minor error.
 - **Cancel:** The running process (cycle run, traverse command, etc.) was aborted. You can resume operation once the error has been cleared.
 - **Emergency stop:** Traverse and the execution of the DIN program were stopped. You can resume operation once the error has been cleared.
 - **Reset:** Traverse and the execution of the DIN program were stopped. Switch off the control for a moment, then restart. Contact your machine manufacturer if the error occurs again.

System Error, Internal Error

If a **system error** or **internal error** occurs, write down all information on the displayed message and inform your machine manufacturer. You cannot correct an internal error. Switch off the control and restart.

Warnings during Simulation

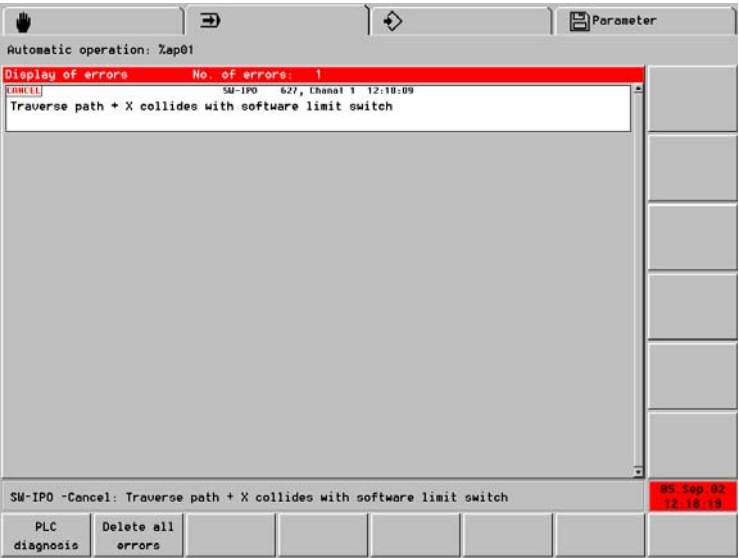
In the event of problems during simulation of an NC program, the CNC PILOT displays a warning in the status line (see "5.1.2 Notes on Operation").

PLC display

The PLC window is used for PLC messages and the PLC diagnosis. Your machine manual provides more detailed information on the PLC window.

To call the PLC window, open the error window with the Error status key and then press "PLC Diagnosis" soft key.

To exit the PLC status display, press the ESC key; to switch to the error window, use the "CNC Diagnosis" soft key.



Soft keys

PLC diagnosis	Switch to PLC display
Delete all errors	Deletes all error messages
CNC diagnosis	Return to error display

2.4 Data Backup

The CNC PILOT stores NC programs, operating-resource data and parameters on the hard disk. Since the possibility of damage to the hard disk due to excessive vibration or shock cannot be eliminated, HEIDENHAIN recommends making regular backup copies of your programs, operating resource data and parameters on a PC.

You can use DataPilot 4290, the WINDOWS "Explorer" or other suitable programs for backing up your data on a PC.

For data exchange and data backup, you can use the **Ethernet interface**. Data exchange is also possible over the **serial interface (RS-232)** (see "10.2 Data Transfer Methods").

2.5 Explanation of Terms

- **Cursor:** In lists, or during data input, a list item, an input box or a character is highlighted. This "highlight" is called a **cursor**.
- **Arrow keys:** The cursor is moved with the "page up/page down" arrow keys or the touch pad.
- **Navigate:** You can move the cursor within a list or an input box to any position you would like to check, change, delete or add to. In other words, you "navigate" through the list.
- **Active/inactive functions, menu items:** Functions or soft keys that currently unavailable are shown dimmed.
- **Dialog box:** Dialog boxes are also called input windows.
- **Editing:** "Editing" is changing, deleting and adding to parameters, commands, etc., within programs, tool data or parameters.
- **Default value:** If the parameters of DIN commands or other parameters are preassigned values, these values are referred to as "default values."
- **Bytes:** The capacity of a storage disk is indicated in "bytes." Since the CNC features a hard disk, the individual program lengths (file sizes) are expressed in bytes.
- **Extension:** File names consist of the actual file name and the "file name extension." The name part and the extension part are separated by ".". The extension indicates the type of file. Examples:
 - "*.NC" DIN programs
 - "*.NCS" DIN subprograms
 - "*.MAS" Machine parameters



3

**Manual Control
and Automatic mode**

3.1 Switch-On, Switch-Off, Reference Run

3.1.1 Switch-On and Traversing the Reference Marks

In the screen dialog line, the CNC PILOT shows you step by step how to proceed when starting the system. The CNC PILOT asks you to select an operating mode.

Whether the reference run is necessary depends on the encoders installed in your machine:

- EnDat encoder: Reference run is not necessary
- Distance-coded encoders: The position of the axes is ascertained after a short reference run
- Standard encoder: The axes move to familiar, machine-based points

"Reference automatic" means that all axes make reference runs. "Reference jog" only one axis does.

Reference automatic (all axes)

Select "Ref – Reference automatic."

"Status of reference run approach" informs you of the current status. Axes that have not been referenced are shown in gray.

Either set the slides that need to find a reference or set "All slides" ("reference automatic" dialog box)



The axis move to find the reference



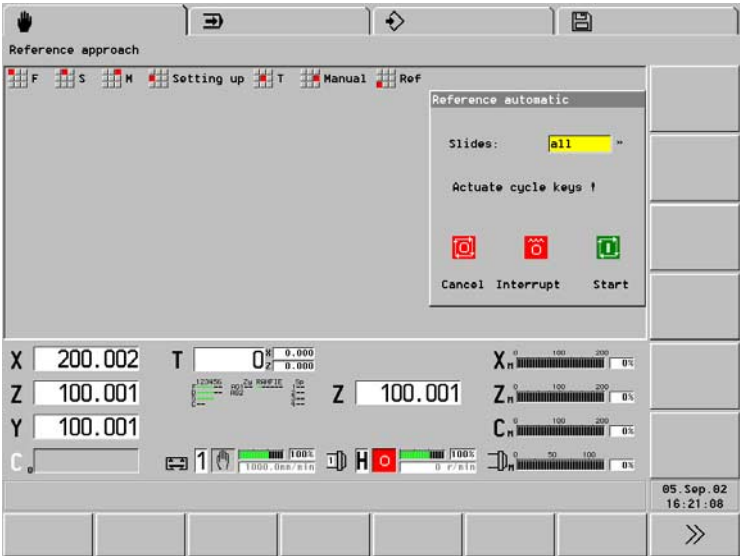
Interrupts the reference run. Cycle start resumes the run.



Cancels the reference run

After completion of the reference run:

- The position display is activated.
- The automatic mode is selectable.



- The **Sequence**, in which the axis make their reference run is defined in machine parameters 203, 253, ..
- Exiting the "Reference automatic" dialog box: Press Cycle stop



The **software limit switches** are active only after you have traversed the reference marks.

Monitoring the EnDat encoders

If your machine is equipped with EnDat encoders, the control saves the axis positions during switch-off. During switch-on, the CNC PILOT compares for each axis the position during switch-on with the position saved during switch-off.

If there is a difference, one of the following messages appears:

- "Axis was moved after the machine was switched off." Check the current position and confirm it if the axis was in fact moved.
- "Saved encoder position of the axis is invalid" This message is correct if the control has been switched on for the first time, or if the encoder or other control components involved were exchanged.
- "Parameters were changed. Saved encoder position of the axis is invalid." This message is correct if configuration parameters were changed.


The cause for one of the messages listed above could be a defect in the encoder or in the control. Please contact your machine supplier if the problem recurs.


Reference jog (single axis)

Select "Ref – Reference jog."

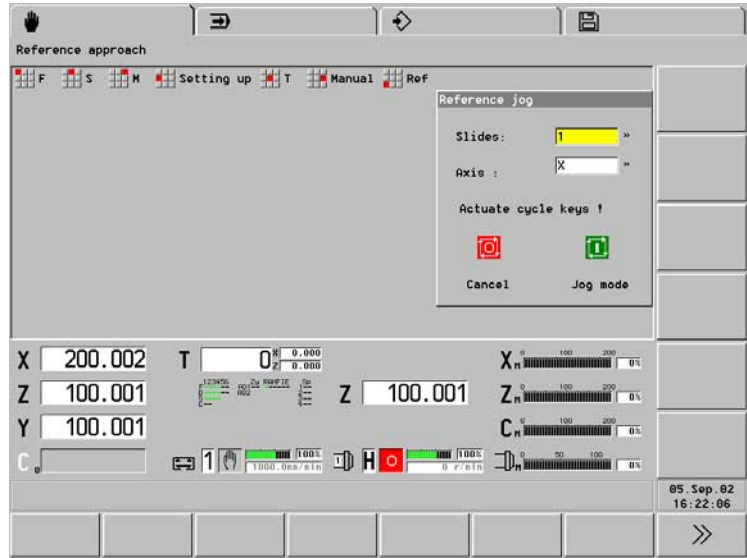
"Status of reference run approach" informs you of the current status. Axes that have not been referenced are shown in gray.


Set slides and axes ("reference jog" dialog box)


 The reference run is continued as long as you keep pressing the key. To interrupt the reference run, release the key.

 Cancels the reference run

- After completion of the reference run:
- The position display is active for the axis that has been referenced.
 - If all axes have been referenced, you can select automatic mode.



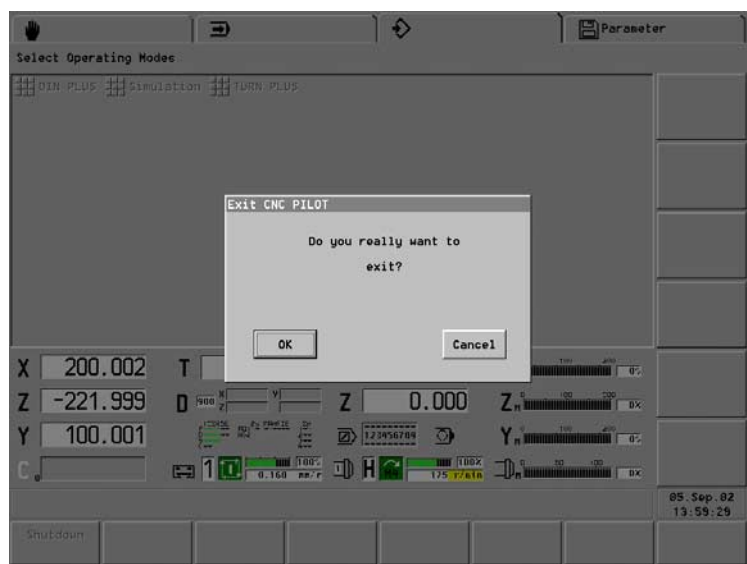
 Exiting the "Reference jog" dialog box: Press cycle stop

 The **software limit switches** are only active after you have traversed the reference marks.

3.1.2 Switch-Off

Shutdown Switching off the CNC PILOT. Confirm the subsequent request with OK. The control is shut down in an orderly manner. After a few seconds, CNC PILOT requests you to switch off the machine.

"Shutdown" is available in the programming and organization modes if no operating mode is selected. Proper switch-off is recorded in the error log file.



3.2 Manual Operating Mode

The Manual control mode offers various functions for setting up the machine, for measuring tool dimensions and for manually machining workpieces.

The **machine display** in the lower section of the screen shows the tool position and further machine data.

Options of operation:

■ **Manual mode of operation**

With the “machine keys” and the handwheel, you can control the spindle and move the axes to machine the workpiece.

■ **Setting up the machine**

Functions for entering the tools being used, setting the workpiece zero point, the tool change position, the protective-zone dimensions, etc.

■ **Measuring tool dimensions**

Functions for measuring the tool by touching the workpiece or by use of measuring devices.

■ **Configuring the screen display**

The CNC PILOT supports various types of machine display.

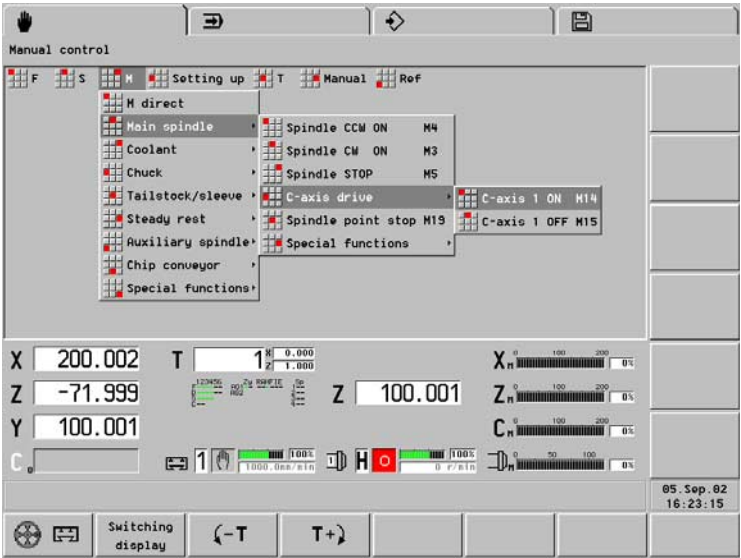


In Automatic mode, the data are entered and displayed in 1 **millimeters** or in **inches**, depending on the setting of the control parameter 1.



Remember: If the machine has **not been referenced**:

- The position display is not valid
- The **software limit switches** are nonoperational.



Soft keys



- Assigns a handwheel to an axis
- Defines the handwheel interpolation factor

Switching display

Switches the machine display



Turret one position backward



Turret one position forward

3.2.1 Entering machine data

"F" (feed rate) pull-down menu:

■ Feed per revolution

- ▶ Select "Feed per revolut."
- ▶ Enter the feed rate in mm/rev (or inches/rev)

■ Feed per minute

- ▶ Select "Feed per minute."
- ▶ Enter the feed rate in mm/min (or inches/min) and press OK.

"S" (spindle speed) pull-down menu:

■ Spindle speed

- ▶ Select "Speed S."
- ▶ Enter the speed in rpm

■ Constant cutting speed

- ▶ Select "V constant."
- ▶ Enter the cutting speed in m/min (or ft/min) and press OK.

■ Spindle point stop

- ▶ To switch to the required spindle, press the Spindle change key.
- ▶ Select "Spindle point stop."
- ▶ Enter position
- ▶ Cycle start: The spindle is positioned
- ▶ Cycle stop: Exit the dialog box

Menu item "T" (Tool):

- ▶ Select "T"
- ▶ Enter the turret position



You can enter a constant cutting speed only for slides with an X axis.



Tool change functions:

- Moving the tool into position
- Offsetting "new" tool dimensions
- Showing the "new" actual values in the position display.

3.2.2 M Commands

"M" (M functions) pull-down menu:

- The M number is known: Select "M direct" and enter the number.
- M menu: To select the M function, use the menu.

After input/selection of the M function:

- ▶ Cycle start: The M function is executed
- ▶ Cycle stop: Exit the dialog box



The content of the M menu depends on the machine. Yours may differ from the example shown here.



3.2.3 ManualTurning Operations

"Manual" pull-down menu:

■ Simple longitudinal and transverse turning operations

- ▶ Select "Constant feed."
- ▶ Select the direction of feed ("Constant feed" dialog box).
- ▶ Control the feed rate with the cycle keys.

■ G functions

- ▶ Select "G function."
- ▶ Enter the G number and the function parameter; press OK.
- ▶ The G function is executed.

The following G functions are permitted:

- G30 – Rear-face machining
- G710 – Adding tool dimensions
- G720 – Spindle synchronization
- G602..G699 – PLC functions

■ Manual NC programs

Depending on the configuration of a lathe, the machine manufacturer can include NC programs supporting the machinist in manually operating the lathe (Example: Switching to rear-face machining). Refer to the machine manual.



With constant speed, a feed rate per revolution must be defined.

3.2.4 Handwheel



Assign the handwheel to one of the principle axes or the C axis, and enter the feed rate or angle of rotation per handwheel increment ("Handwheel axes" dialog box).

The handwheel assignment and speed ratio are shown in the machine display (the axis letter and the decimal place of the handwheel traverse ratio are marked).

To cancel the handwheel assignment, press the "Handwheel" soft key with the opened dialog box.


The handwheel assignment is canceled by:

- Switching to another slide.
- Changing the operating mode.
- Pressing an axis direction key.
- Selecting the handwheel assignment again.








3.2.5 Spindle and Axis Direction Keys

The keys of the machine operating panel are used for machining a workpiece manually and for special functions such as positioning or determining compensation values (actual position capture, scratching, etc.).
To activate tools, define the spindle speed and feed rate, etc., use the menus.










To move the slide diagonally, press the X and Z-axis direction keys simultaneously.

Spindle keys

		Switch the spindle on in M3/M4 direction.
		Jog the spindle in M3/M4 direction. The spindle rotates as long as the key is held. Jog speed: machine parameters 805, 855, ...
		Spindle stop



Axis direction keys (jog keys)

		Move slide in X direction.
		Move slide in Z direction.
		Move slide in Y direction.
		To move the slide in rapid traverse: Simultaneously press the rapid traverse key and the axis direction key. Rapid traverse velocity: Machine parameters 204, 254, ...

3.2.6 Slide/Spindle change key

- On lathes with more than one slide, the **axis direction keys** control the selected slide.
 - Selection of the slide: Slide change key
 - Display of the selected slide: Machine display
- On lathes with more than one spindle, the **spindle keys** on the selected spindle.
 - Select the spindle: Spindle change key
 - Display of the selected spindle: Machine display.
- For **setup functions** referring to one slide or spindle (workpiece zero point, tool change point, etc.), you specify the slide/spindle with the slide/spindle change key.
- The machine display usually contains display elements for spindle and slide. To switch between these elements, use the Slide/Spindle change key (see 3.6 "Machine Display").

Slide/Spindle change key

	Switch over to the next slide
	Switch over to the next spindle

3.3 Tool Lists, Tool Life Management

The tool list (turret table) indicates the current tool carrier assignment. To compile a tool list, enter the ID numbers of the tools.

You can use the entries in the TURRET section of the NC program to set up the tool list. The "Compare list" and "Accept list" functions refer to the NC program last interpreted in automatic mode.

Tool life data

Apart from ID numbers and tool type descriptions, the tool list includes data for tool life management:

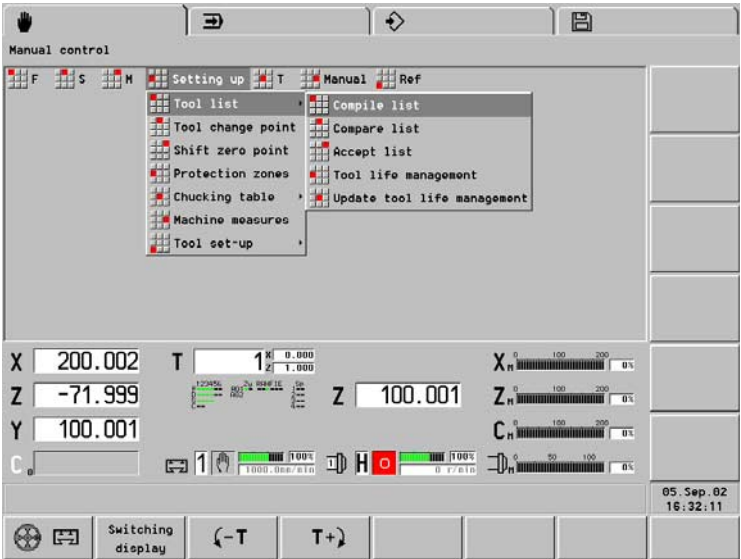
- **Status**
Shows the remaining tool life/quantity.
- **Ready for use**
When the tool life has expired/the defined number of parts has been produced, the tool is "not ready for use" any longer.
- **Atw (replacement tool)**
If a tool is "not ready for use," a replacement tool is inserted.

Simple tools

With the setup functions, you can only enter tools registered in the database. If the NC program uses "simple tools," proceed as follows:: Run an interpretation of the NC program; the CNC PILOT automatically updates the tool list.h

- If the positions in the tool list are occupied by "old" tools, the confirmation request - "Update tool list?" - appears. The tools are only entered after you have confirmed the request.

Tools that are not registered in the database are identified by the code "_AUTO_xx" (xx:T number), and not by an ID number.



- The parameters of simple tools are defined in the NC program
- The tool life data are evaluation only if the tool life management is **active**.



Danger of collision

- Compare the tool list with the current tool carrier assignment and check the tool data **before** running a program.
- The tool list and the dimensions of the tools entered must correspond to the actual facts, because the CNC PILOT uses the data for slide movements, protective-zone monitoring, etc.

3.3.1 Setting Up a Tool List

A tool list can also be set up without using an NC program.


Enter a new tool

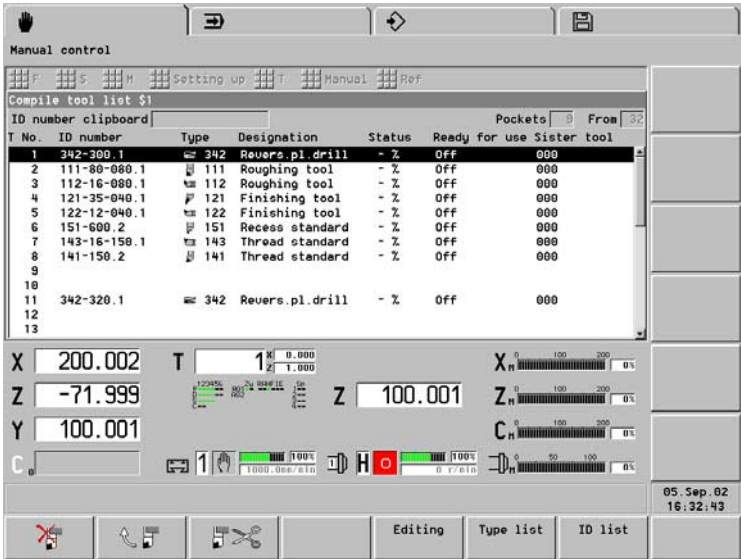
- Select "Setting up -Tool list - Compile list"
- Select the tool location
- ENTER (or INS key) – opens the setup dialog box
- Enter the ID number

Take the tool from the database




- Type list Enter the tool type – the CNC PILOT displays all tools of this type mask
- ID list Enter the ID number – the CNC PILOT displays all the tools of this ID mask
- Select the tool
- Insert Take the tool from the database
- Esc Exit the tool database

Delete the tool

- Select "Setting up -Tool list - Compile list"
- Select the tool location
-  or the DEL key deletes the tool




Soft keys

-  Delete tool
-  Take the tool from the "ID number clipboard"
-  Delete the tool and place in the "ID number clipboard"
- Editing Edit the tool parameters
- Type list Entries in the tool database - sorted by tool type
- ID list Entries in the tool database - sorted by tool ID number


Changing the tool pocket

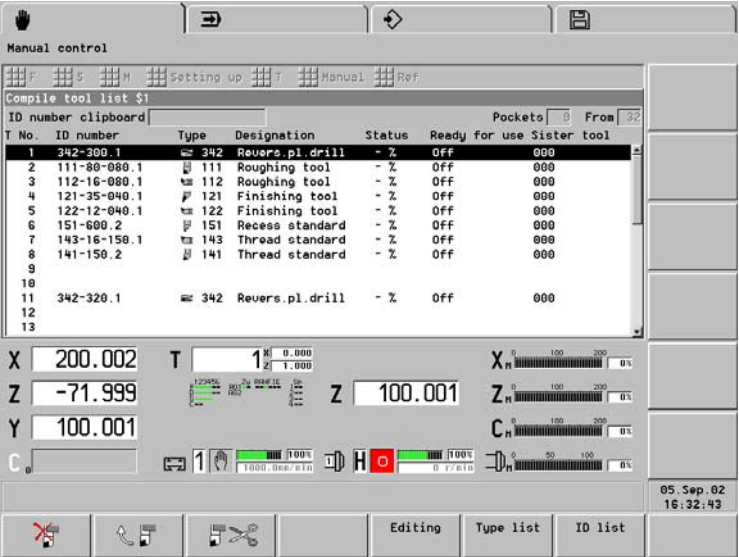
Select "Setting up -Tool list - Compile list"

Select tool location

 Deletes the tool and saves it in the "ID number clipboard"

Select a new tool location

 Take the tool from the "ID number clipboard"
If the location was occupied, the previous tool is taken into the clipboard.



3.3.2 Comparing a Tool List with an NC Program

The CNC PILOT compares the current tool list with the entries in the NC program last translated in automatic mode.

Comparing a tool list

Select "Setting up -Tool list - Compare list" The CNC PILOT shows the current contents of the tool list and marks deviations from the programmed tool list.

Select marked tool location

Nominal-actual comparison

Press ENTER (or INS key). The CNC PILOT opens the "nominal-actual comparison" dialog box.

Confirm nominal

Accept the ID number of the "nominal tool" in the tool list or

Type list

Look for the tool in the database


ID list

The CNC PILOT shows the following tools marked:

- Actual tool ≠ nominal tool
- Actual – not occupied; nominal – occupied

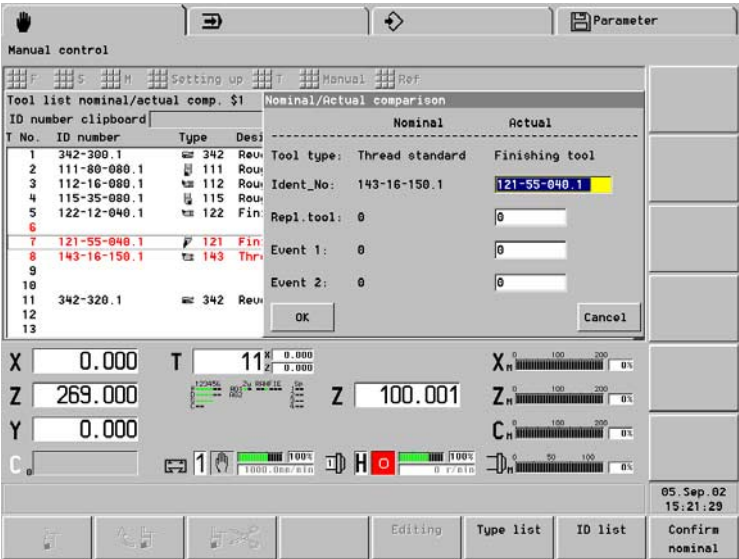
The entries in the TURRET section are considered nominal tools (reference: TURRET section of the NC program most recently interpreted in Automatic mode).

Tool locations that are not assigned in the NC program cannot be selected.




Danger of collision


- Tool pockets that are occupied but, according to the NC program, are not needed, are **not** marked.
- The CNC PILOT compensates the tool actually entered – even if it does not match the nominal assignment.




Soft keys



Delete the tool



Take the tool from the "ID number clipboard"



Delete the tool and place in the "ID number clipboard"

Editing

Edit the tool parameters

Type list

Entries in the tool database - sorted by tool type

ID list

Entries in the tool database - sorted by tool ID number

Confirm nominal

Accept the ID number of the "nominal tool" in the tool list

3.3.3 Transferring the Tool List from an NC Program

The CNC PILOT transfers the new tool assignment from the TURRET section (reference: the NC program last interpreted in Automatic mode).

Transferring the tool list

Select "Setting up -Tool list -Accept list"

Depending on the previous turret assignment, the following might occur:

- **Tool not used**
The CNC PILOT enters the new tools in the tool list. Positions that were occupied in the old tool list, but are not used in the new list, are retained. If a tool shall remain in the tool carrier, no further action is required; if not, delete the tool:

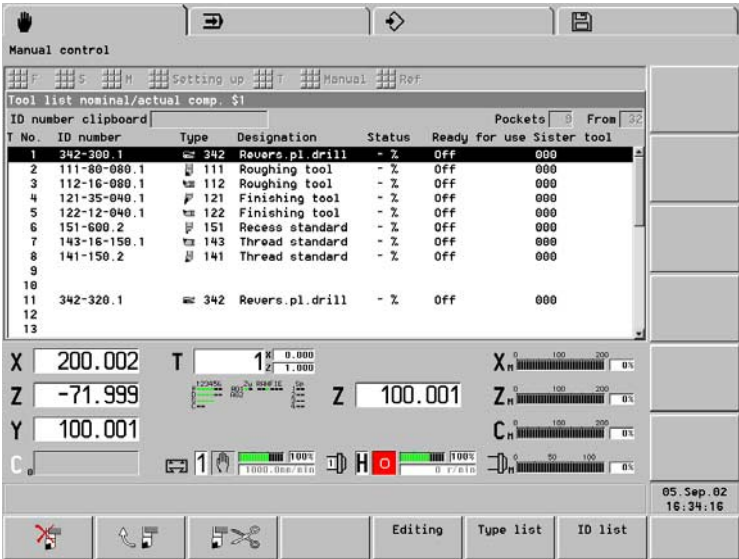
- **Actual tool location differs from location in tool list**
A tool is **not** entered when its newly assigned location differs from the location specified in the tool list. The CNC PILOT displays a message indicating this error. Change the tool location.

As long as a tool position differs from the nominal assignment it remains highlighted.



Danger of collision

- Tool locations that are occupied but, according to the NC program, are not needed, are kept.
- The CNC PILOT compensates the tool actually entered – even if it does not match the nominal assignment.



Soft keys



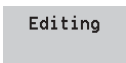
Delete the tool



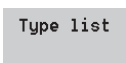
Take the tool from the "ID number clipboard"



Delete the tool and place in the "ID number clipboard"



Edit the tool parameters



Entries in the tool database - sorted by tool type



Entries in the tool database - sorted by tool ID number

3.3.4 Tool Life Management

The tool life management allows you to define the sequence of exchange and declare the tool to be ready for use. The tool life/quantity is defined in the tool database (see section "8.1.7 Multiple Tools, Tool Life Monitoring").

The "Tool life management" dialog box is used both for entering and displaying the tool life data.

You can use the variable-programming function in your NC program to evaluate sequential events that you enter in "Event 1" and "Event 2" (see section "4.15.2V Variables").

Tool life management parameters

- **Repl. tool** (replacement tool): T number (turret position) of the replacement tool
- **Event 1**: Sequential event that is triggered when the life of a tool has expired/a tool has produced the defined quantity – Event 21..59
- **Event 2**: Sequential event that is triggered when the life of the last tool of the interchange chain has expired/the tool has produced the defined quantity – Event 21..59
- **Ready for use**: Set the tool to "ready for use" or "not ready for use" (applies to tool life management only).

Entering the tool life parameters

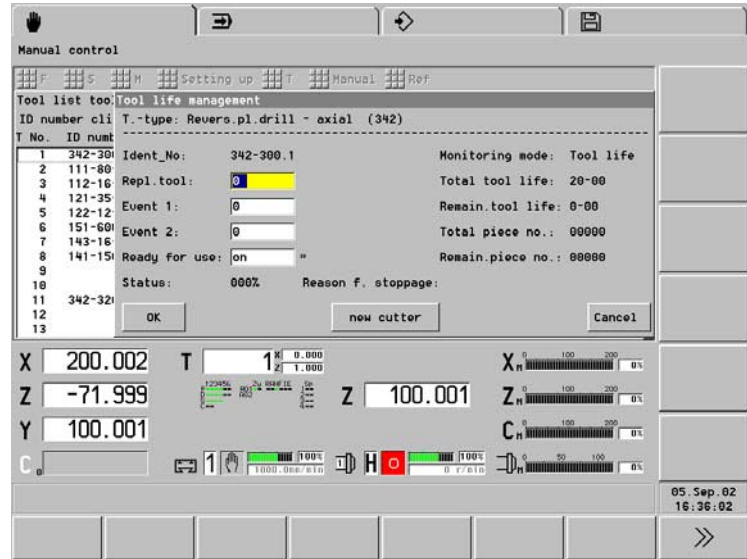
Select "Setting up – Tool list – Tool life management"; the CNC PILOT displays the tools entered.

Select the tool location.

Press ENTER – the CNC PILOT opens the "Tool life management" dialog box.

Enter the replacement tool and the tool life parameters; press OK.

"New cutter" sets the tool life/quantity to the value programmed in the database and sets the tool to **ready for use**.



Update tool life management data

Select "Setting up – Tool list – Update tool life management."

Confirm the confirmation request with OK; the CNC PILOT sets the tool life/quantity to the value defined in the database and sets all tools in the tool list to **ready for use**.

The CNC PILOT displays the "Tool list - tool life management" for inspection.

Application example: The cutting edges of all tools used have been replaced. Part production is to be continued, using the tool life management function.

3.4 Setup Functions

3.4.1 Defining the Tool Change Position

With the ISO command G14, the machine slide moves to the **tool change point**. Always program the tool change point as far from the workpiece as possible to allow the turret to rotate to any position.

Defining the tool change position

For more than one slide: Define the desired slide (with the Slide change key)

Select "Setting up –Tool change point."

The CNC PILOT displays the currently valid position in the "Set tool change point" dialog box.

Entering the tool change point

Enter a new position

Capture tool change point

Move slide to the tool change position

Confirm position


Confirms the slide position as tool change point

or

Confirm X


Confirms the position of individual axes

The tool change point is managed in the setup parameters (Select: „Act. Para – Setting up (menu) – Tool change point – ..“).

 The coordinates of the tool change position are entered and displayed as distance between machine datum and tool carrier datum. Since these values are not shown in the position display, it is advisable to move to the tool change point and "capture" the position.



Soft keys



■ Assigns a handwheel to an axis
■ Defines the handwheel interpolation factor

Switching display

Switch the machine display

F

Enter the feed per revolution

S

Enter the constant surface speed

M

Enter the M function

Confirm X

Accept the axis position as tool change point(or Y or Z axis)

Confirm position

Accept the slide position as tool change point

3.4.2 Shifting the Workpiece Datum

Shifting the Workpiece Datum

For more than one slide: Define the desired slide (with the Slide change key)

Position the tool

Select "Setting up - Shift zero point."

The "Shift zero point" dialog box displays the current workpiece zero point.

Enter the workpiece zero point
Enter a "zero point shift"

Contact position = tool zero point

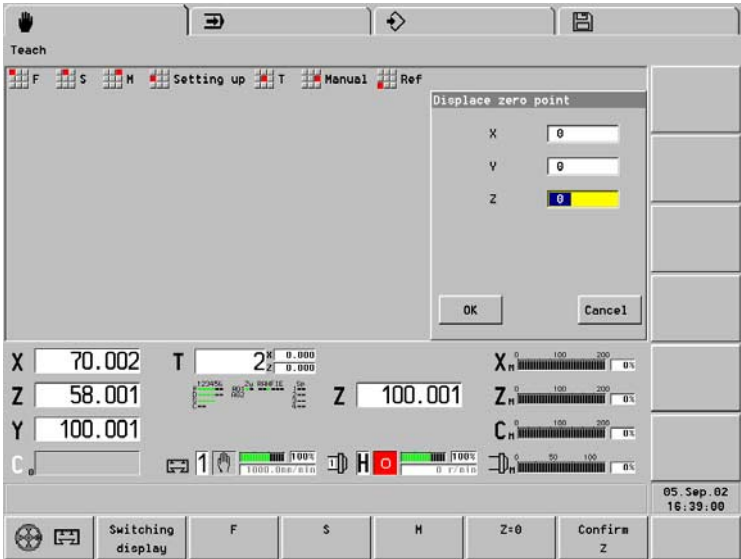
- ▶ Touch the end face with the tool
- ▶ Accept the tool contact position as workpiece zero point

Workpiece zero point relative to the contact position

- ▶ Touch the end face with the tool
- ▶ Accept the tool contact position
- ▶ Enter the measured value (distance of the tool contact position from the workpiece zero point)

The tool zero point is managed in the setup parameters (Select: „Act. Para – Setting up (menu) – Tool zero point – ..“).

- The "displacement" is with respect to the machine zero point.
- You can also offset the workpiece zero point for the X and Y axes.



Soft keys

- Assigns a handwheel to an axis
- Defines the handwheel interpolation factor
- Switching display** Switches the machine display
- F** Enter the feed per revolution
- S** Enter the constant surface speed
- M** Enter the M function
- Z=0** Define the Z position as tool zero point (or X or Y position)
- Confirm Z** Specify the tool zero point relative to the current Z position (or X or Y position)

3.4.3 Defining the protection zone

Defining the protection zone

Insert any tool (T0 is not permitted).

Select "Setting up - Selection zones"

Enter the protection zone parameters
Enter the limit values.

Capturing the protection zone parameters per axis


- For each input box:
- ▶ Select the input field
 - ▶ Position the tool to the protection zone limit

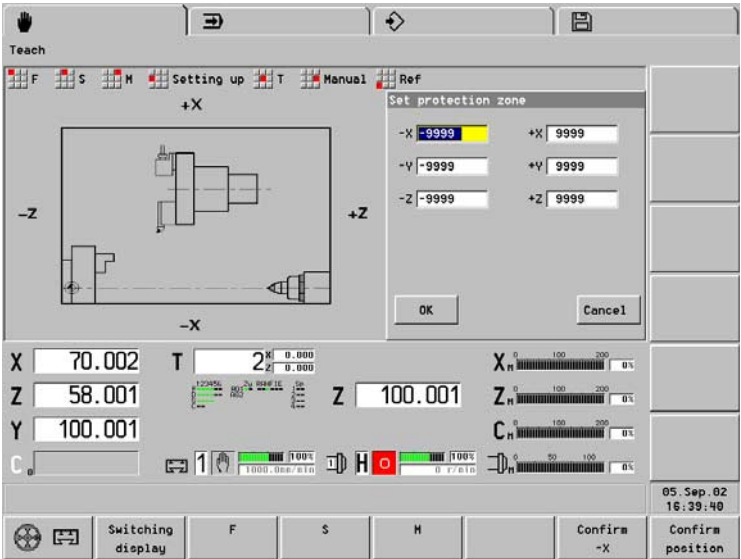
Confirm -X Accept the axis position as protection zone parameter

Capturing positive/negative protection zone parameters


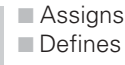
- ▶ Select any positive or negative input field
- ▶ Position the tool to the protection zone limit

Confirm position Accept all positive/negative axis positions

 The parameters serve for protection zone monitoring - not as software limit switches.
Protection zone parameters:
■ are expressed with respect to the machine zero point
■ are managed in the machine parameters 1116, 1156, ...
■ X value are radius dimensions
■ 99999/-99999 means: no monitoring of this side of the protection zone



Soft keys

-  Assigns a handwheel to an axis
-  Defines the handwheel interpolation factor

Switching display Switch the machine display

F Enter the feed per revolution

S Enter the constant surface speed

M Enter the M function

Confirm -X Accept the X position as "protection zone -X" parameter (or +X, -Y, +Y, -Z, +Z position)

Confirm position Accept the axis positions as positive/negative protection zone parameter

3.4.4 Setting up the Chucking Table

The **chucking table** is evaluated by the concurrent graphics.

Setting up the chucking table

Select "Setting up – Chucking table – Main spindle (or Tailstock)

Select the ID number from the chucking database

Chucking equipment for spindles

The definition of the clamping form ("Grip. form") presupposes the definition of the chuck jaws. Set the clamp form by soft key – it is graphically illustrated.

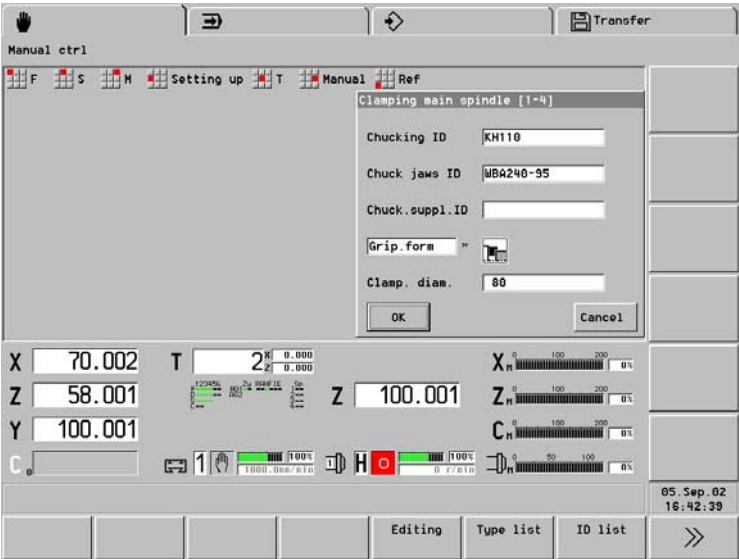
To switch to the chucking assignment of further spindles, press the Page Up/Page Dn keys.

Parameters for "spindle x" (main spindle, spindle 1, ..)

- Chucking ID (identification number): Reference to database.
- Chuck jaws ID (identification number): Reference to database.
- Chuck supplement ID (identification number): Reference to database.
- Clamp form (for chuck jaws): Define internal/ external chucking and the level of chuck jaws used
- Clamping diameter: The diameter at which the workpiece is clamped. (Workpiece diameter when clamped externally; inside diameter when internal clamping is used)

"Tailstock" parameters

- Sleeve center ID (identification number): Reference to database.



Soft keys

Editing	Edit the chucking equipment parameters
Type list	Entries in the chucking database – sorted by chuck type
ID list	Entries in the chucking database – sorted by chuck ID number
>>	"Continue" – Define the clamp form

3.4.5 Setting up Machine Dimensions

You can evaluate machine dimensions in the variable programming of the NC program.

The "Set machine dimensions" function accounts for the dimensions 1..9 and the "configured axes" for each dimension.

Setting up machine dimensions

Select "Setting up - Machine dimensions."

Enter the machine dimension number

Enter the machine dimensions

Enter the values ("Set machine dimension x" dialog box).

Capturing a single machine dimension

- ▶ Select the input field
- ▶ Move the axis to the desired position

Confirm
X

Confirm the axis position as machine dimension (orY or Z position)

Capturing all machine dimensions

- ▶ Move the slides to the desired positions

Confirm
position

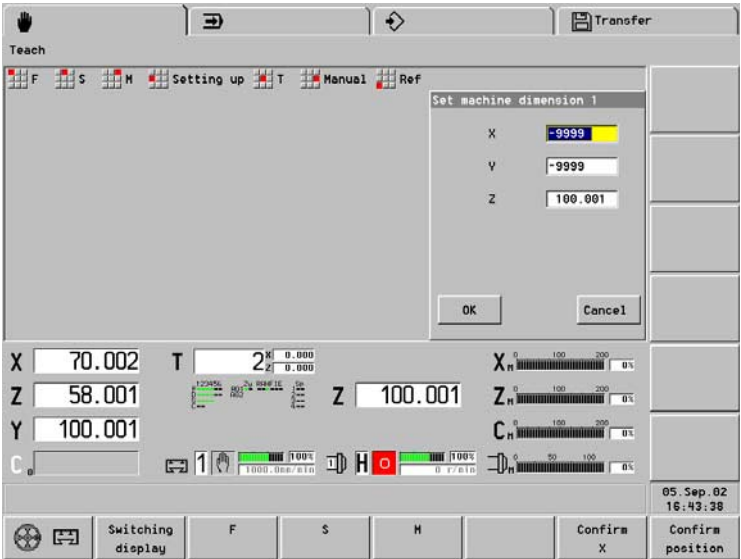
Confirm the axis positions of the slides as machine dimensions

- ▶ OK – enter the next machine dimension
- ▶ Cancel – Exit the machine dimension setup

Machine dimensions are managed in machine parameter 7.



Machine dimensions are given with respect to the machine zero point.



Soft keys



- Assigns a handwheel to an axis
- Defines the handwheel interpolation factor

Switching
display

Switch the machine display

F

Enter the feed per revolution

S

Enter the constant surface speed

M

Enter the M function

Confirm
X

Accept the axis position as machine dimension X (orY or Z axis)

Confirm
position

Accept the axis positions of the slides as machine dimensions

3.4.6 MeasuringTools

Define the type of tool measurement in machine parameter 6:

- 0: Contact with tool
- 1: Measure with touch probe
- 2: Measure with measuring optics

MeasuringTools

Position the tool

Select "Setting up -Tool setup –Tool measuring." The "Tool measuringT..." dialog box indicates the current tool dimensions.

Enter the machine dimensions

Enter the dimensions

Find the tool dimensions by touching the workpiece with the tool

- ▶ Select the input field "X"
- ▶ Touch off the diameter, retract in Z direction
- Confirm X** Confirm the diameter as measured value
- ▶ Select the Z input field
- ▶ Touch the face with the tool, then retract in the X direction
- Confirm Z** Confirm the Z position of the tool as the measured value

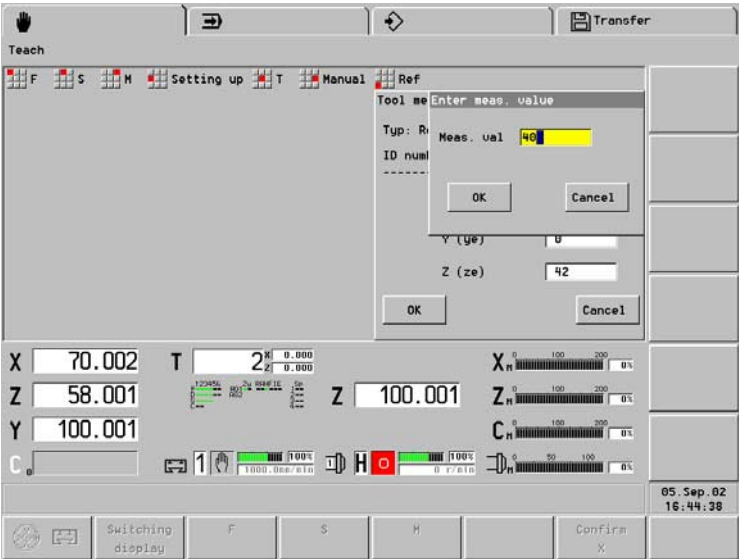
Measuring tools with the touch probe

- for each input field:
- ▶ Select the X/Z input field
 - ▶ Move the tool tip in X/Z direction to the probe; the CNC PILOT saves the X/Z dimension
 - ▶ Retract the tool – retract the touch probe

Measuring the tools with measuring optics

- For each input field
- ▶ Select the X/Z input field
 - ▶ Align the tool point in the X/Z direction with the cross hairs

Confirm X Accept the value



Soft keys

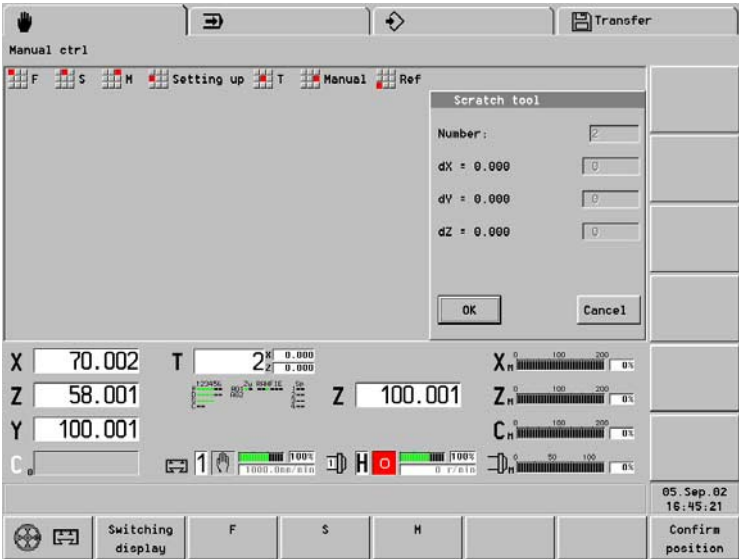
	<ul style="list-style-type: none"> ■ Assign a handwheel to an axis ■ Define the handwheel interpolation factor
Switching display	Switch the machine display
F	Enter the feed per revolution
S	Enter the constant surface speed
M	Enter the M function
Confirm X	Accept the X position as measured value X (or Y or Z position)

- ▶ The entries in the "Enter measured value" dialog box are given with respect to the workpiece zero point.
- ▶ The compensation values of the tool are deleted.
- ▶ The measured tool dimensions are entered in the database.

Determining tool-compensation values

- Move the tool into position
- Select "Setting up -Tool setup –Tool compensation"
- Assign the handwheel to the X axis – move the tool by the compensation value
- Assign the handwheel to the Z axis – move the tool by the compensation value

Confirm position The CNC PILOT loads the compensation values



Soft keys

- Assign a handwheel to an axis
- Define the handwheel interpolation factor

Switching display Switch the machine display


F Enter the feed per revolution

S Enter the constant surface speed

M Enter the M function

Confirm position Accept the tool compensation values

3.5 Automatic Mode of Operation

 In Automatic mode, the data are entered and displayed in **1 millimeters** or in **inches**, depending on the setting of the control parameter 1. The setting in the "program head" of the NC program governs the execution of the NC part program - it has no influence on operation or display.

3.5.1 Program Selection

The CNC PILOT interprets the NC program before it can be activated with Cycle Start. "#-Variables" are entered during the translation process. A "restart" prevents a new translation, while a "new start" forces a new translation.

Program selection

- ▶ Select "Prog – Program selection"
- ▶ Select the NC program
 - The NC program is loaded without previous translation, if:
 - No changes were made in the program or the tool list.
 - The lathe was not switched off since the program was last selected.

Restart


- ▶ Select "Prog – Restart"
- The last active NC program is loaded without without previous translation, if:
 - No changes were made in the program or the tool list.
 - The lathe was not switched off since the program was last selected.

New start

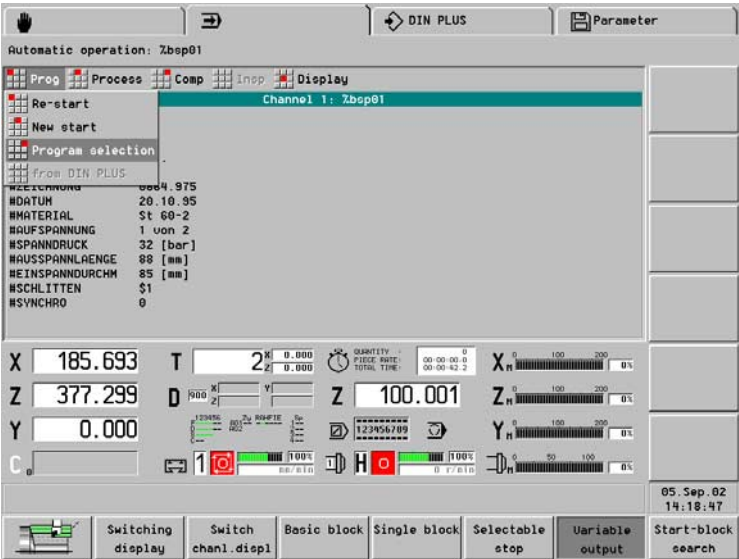
- ▶ Select "Prog – New start"
- The NC program is loaded and **translated**. (Use for starting an NC program containing # variables.)

From DIN PLUS

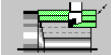
- ▶ Select "Prog – From DIN PLUS"
- The NC program selected in DIN PLUS is loaded and **translated**.



- If the "turret table" of the NC program is not the currently valid one, there is a warning.
- The name of the NC program is retained until you select another program – even if the lathe was switched off in the meantime.



Soft keys

	Switch to "graphic display"
Switching display	Switch the machine display
Switch chanl. displ	Define block display for more channels
Basic block	Display basic blocks (individual paths of traverse)
Variable output	Suppress/permit variable output
Single block	Set single block mode
Selectable stop	Program stop at M01 (optional stop)
Start-block search	Run a start-block search

3.5.2 Defining a Start Block

Defining a Start Block


Start-block search Activating a start block search


Position the cursor on the start block. (The soft keys support your search.)

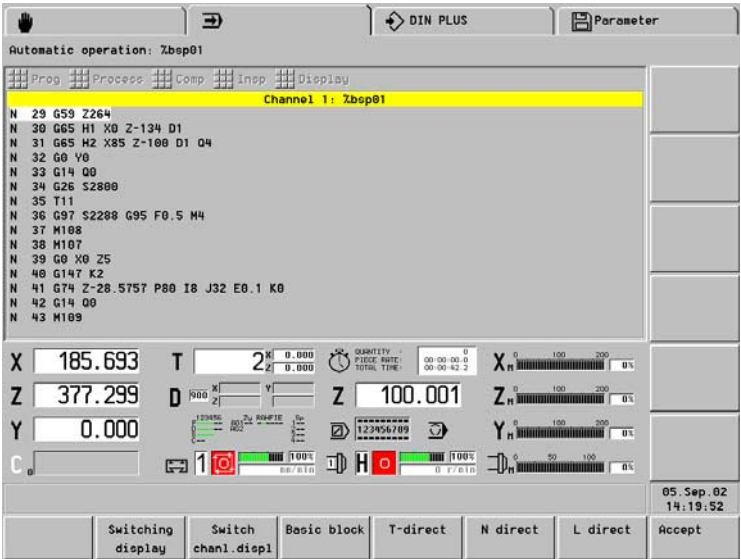
Accept The CNC PILOT switches back into automatic mode and jumps to the start block.

 Start the NC program with the selected NC block.

Esc Exit the start block search without default start block.

-  ■ Select a suitable start block. If program run is started by a specific start block, the CNC PILOT automatically provides all the programmed and essential data for this NC program (excluding interchange of the correct tool and paths of traverse).
- On machines with more than on slide, select a suitable start block on **all** slides before you press the Accept soft key.

 **Danger of collision**
■ If the start block includes a T command, the CNC PILOT first rotates the turret.
■ The first traversing instruction is executed from the current tool position.



Soft keys

- | | |
|----------------------------|---|
| Switching display | Switch the machine display |
| Switch chan1. displ | Define block display for more channels |
| Basic block | Display basic blocks (individual paths of traverse) |
| T-direct | Preset the T number – the cursor is positioned with this T number on the next T command |
| N direct | Preset the T number – the cursor is positioned to the block number |
| L direct | Preset the L number – the cursor is positioned with this L number to the next subprogram call |
| Accept | Run a start-block search |

3.5.3 Program Sequence Modification

Skip levels:

- The program blocks which are marked as active skip levels are **not** executed when the program is being carried out.
- Skip levels: 0..9
- For multiple skip levels, enter a sequence of digits
- Deactivate the skip level: No entries in "Level No."

Operation:

- Select menu item "Process - Skip level"
- Enter the level number

Quantity

- Counting range: 0..9999
- Quantity = 0: Production without quantity limitation; the counter is increased by one after each program run.
- Quantity > 0: The CNC PILOT produces the defined quantity; the counter is reduced by one after each program run.
- Quantity counting is retained even if the machine has been switched off in the meantime.
- When an NC program is activated with "Program selection," the CNC PILOT resets the quantity counter.
- When a program has completed a production lot, the NC program cannot be restarted by the Cycle Start key. To start the NC program again, press "Re-start."

Operation:

- Select menu item "Process – Quantity"
- Enter the quantity

V variables

- The "V variables" dialog box serves for input and display of variables.
- V variables are defined at the beginning of the NC program. The meaning is specified in the NC program.

Operation:

- Select the menu item "Process – V variables" – the CNC PILOT shows the variables defined in the NC program
- Press "Edit" if you wish to change the variables

Continued ►

Status of skip levels

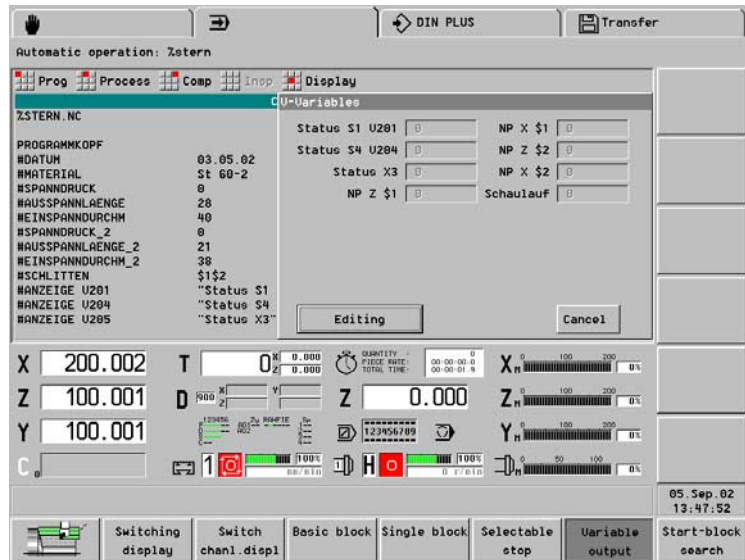
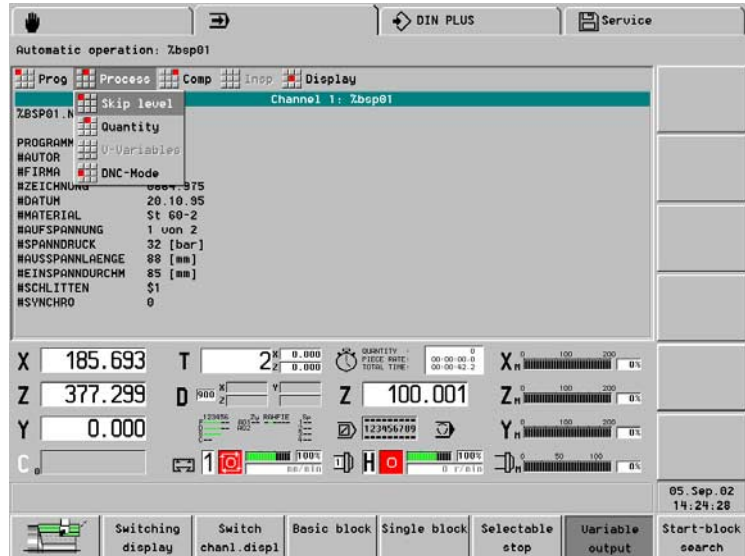
Display field:



Markings:

- Upper row: entered skip levels
- Lower row: the skip levels detected by the "block execution" (active skip levels)

The CNC PILOT accounts for activated/deactivated skip levels after approx. 10 blocks (reason: block scan during the execution of NC blocks).



Single block

Single-block mode
Only **one** NC command (basic block) is executed at a time. The CNC PILOT then goes into the "cycle stop" condition. The subsequent blocks are started with Cycle Start.

Selectable stop


Optional STOP
The CNC PILOT stops at the M01 command and goes into the cycle stop condition. Cycle start resumes the program run.

Feed rate override F% (0% .. 150%)
Feed rate override is controlled manually with a knob on the machine operating panel. The machine display shows the current feed rate override.

Spindle speed override S% (50% .. 150%)
The spindle speed override or the reset to the programmed speed is controlled with the keys of the machine operating panel. The machine display shows the current spindle speed override.

3.5.4 Compensation

- **Tool compensation**
 - ▶ Select "Comp -Tool compensation"
 - ▶ The CNC PILOT enters the T number and current compensation values of the active tool. You can enter a different T number.
 - ▶ Enter the compensation values
 - ▶ Values entered here are added to the existing compensation values.




Tool compensation:


- Become effective when the next traverse starts.
- Are transferred to the database.
- Values of max. 1 mm can be entered.


Status of optional stop


Optional stop off 

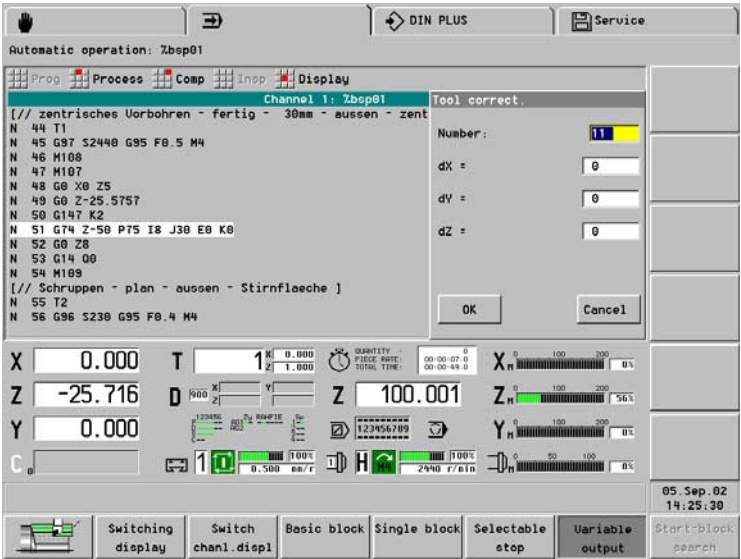
Optional stop on 

Keys for spindle speed override

 Rotational speed to 100% (of the programmed value)

 Increase speed by 5%

 Reduce speed by 5%



Additive compensation

- ▶ Select "Comp - Additive compensation."
- ▶ Enter the compensation number (901 to 916); the CNC PILOT displays the current compensation values.
- ▶ Enter the compensation values
- ▶ Values entered here are added to the existing compensation values.



Additive compensation:

- are activated with G149 ..
- are managed in setup parameter 10
- can be changed by no more than 1 mm

3.5.5 Tool Life Management

- ▶ Select "Comp - Tool life management."
- ▶ This tool list with the current tool life data is displayed
- ▶ Select the tool
- ▶ ENTER opens the "Tool life management" dialog box
 - Set to "ready for use" - or
 - update the tool life data with a "new cutter."

3.5.6 Inspection Mode

This function interrupts the program sequence, checks and corrects the "active tool," inserts a new cutting edge and continues the NC program from the point of interruption.

The inspection cycle is executed as follows:: Interrupt the program sequence and retract the tool. Check the tool, and replace the cutting edge if necessary.n Return the tool.

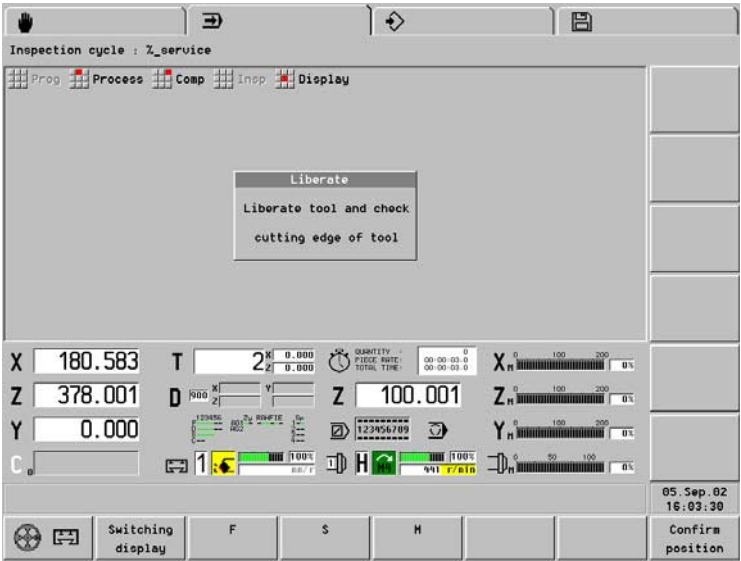
- Cutting edge OK: Continue the automatic program run.
- New cutting edge: Define the compensation values by "scratching," and continue the automatic program run.

When the tool is retracted, the CNC PILOT stores the first five traverse movements. Each change in direction corresponds to a path of traverse.

The NC program run can be continued **before** the point of interruption. Specify the distance to the interruption point. If the value is greater than the distance between the start of the NC block and the interruption point, program sequence begins at the start of the interrupted NC block.



- During the inspection process you can turn the turret, press the spindle keys, etc.
- If the turret was turned, the return motion program inserts the "correct" tool.
- When changing the cutting edge, select the compensation values so that the tool stops **in front of** the workpiece.
- In the cycle stop condition you can interrupt the inspection cycle with ESC and switch to "Manual control."



Inspection mode



Interrupt the program run

Select "Insp(ection)"

To retract the tool, use the axis-direction keys.

If necessary, swivel the turret.

Inspect the tool; if necessary, replace it.

Confirm position Conclude the inspection process - the CNC PILOT loads the return motion program ("_SERVICE").

The "Tool compensation" dialog box appears. Enter the compensation values, and confirm with OK.
If you are using a **new cutting edge**, modify the tool compensation so that the tool - when returning - comes to a stop **in front of** the workpiece.

If necessary, activate the spindle.



Starts the return motion program.

Continued ►

Inspection mode – continued

The "Scrambled takeoff on restart?" dialog box appears; Enter Yes/No, and press OK.

Scrambled takeoff – Yes:

The "Start **from** interruption point (IP) / **before** interruption point" dialog box appears.

- From UP: No further dialog box.
- Before UP: Specify the distance from the point of interruption to the starting point of the tool (Dialog "Distance from interruption point").

The return motion program positions the tool on/ before the interruption point and continues the program **without stopping**.

The inspection cycle has been completed.

Scrambled takeoff – NO:

The "Start **from** interruption point (IP) / **before** interruption point" dialog box appears.

- From UP: No further dialog box.
- Before UP: Specify the distance from the point of interruption to the starting point of the tool (Dialog "Distance from interruption point").

The return motion program positions the tool on/ before the interruption point and **stops**.

Application example: Cutting edge has been replaced.

Select "Insp(ection)" again.

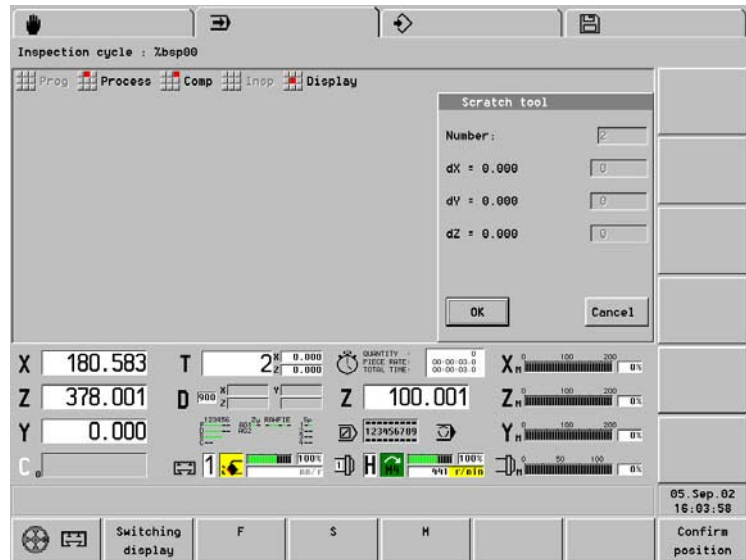
The "Scratch with tool" dialog box opens (for information)

Assign the handwheel to the X/Z axis and "scratch" the workpiece.

Press "Accept value" to save the compensation values defined per handwheel.



The program run continues



3.5.7 Block Display

Block display – basic block display

The **block display** lists the NC blocks according to the programmed sequence. The **basic-block display** shows the individual paths of traverse – the cycles are “resolved.” The numbering of the basic blocks is independent of the programmed block numbers.

In the block display and basic-block display, the cursor is located on the block being executed.

Channel display

For lathes with several slides (channels), you can activate block display for up to 3 channels.

Basic block

Basic block on/off

Switch
chan1.displ

Switching the channel display

To add a channel, press the soft key again; in the block display, only channel 1 is shown.

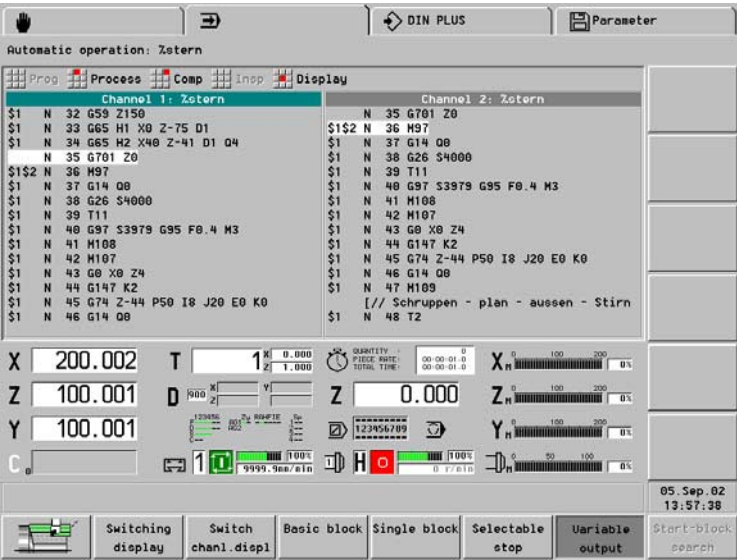
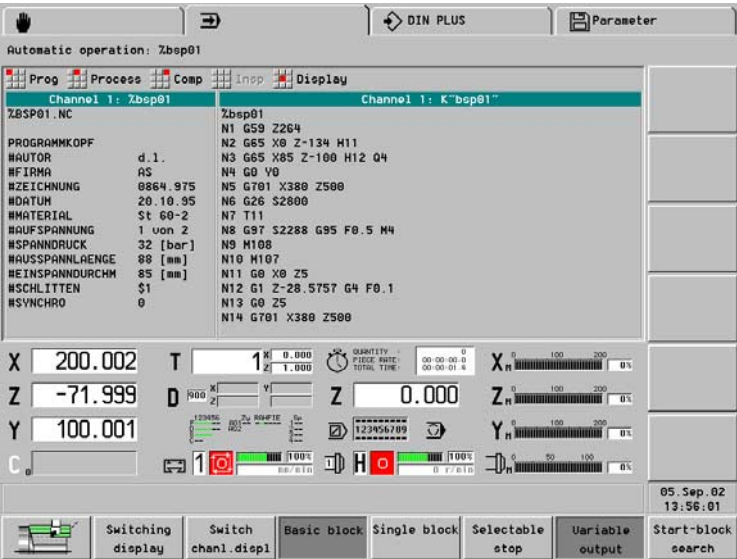
Variable
output

Variable output

Pressing the soft key enables the variable soft key output (with PRINTA). Otherwise the variable output is suppressed.

Menu item “display - ...”

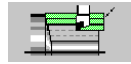
- **Font size:** Enlarges/reduces the characters in the block display
- **Load monitoring** – see “3.7.2 Production under Load Monitoring”



3.5.8 Graphic Display

The "Automatic graphics" function displays the programmed blank and finished part and the paths of traverse. This enables process control of non-visible areas during production and provides an oversight of production status, etc.

All machining operations, including milling, are depicted in the turning window (XZ view).

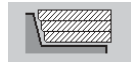


Activate the graphic – if the graphic was already active, the screen is adapted to the current machining status.



Return to block display

Settings:



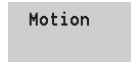
Line: Each tool movement is represented as a line, referenced to the theoretical tool tip.

Cutting path: depicts the surface covered by the "cutting area" of the tool with hatch marks. You can see the area that will actually be machined, with the geometry of the respective tool already accounted for (see "5.1 Simulation Mode of Operation").



The white dot: (small white rectangle) represents the theoretical tool point.

Tool: The tool contour is depicted. (Precondition: Proper tool description in the tool database.)



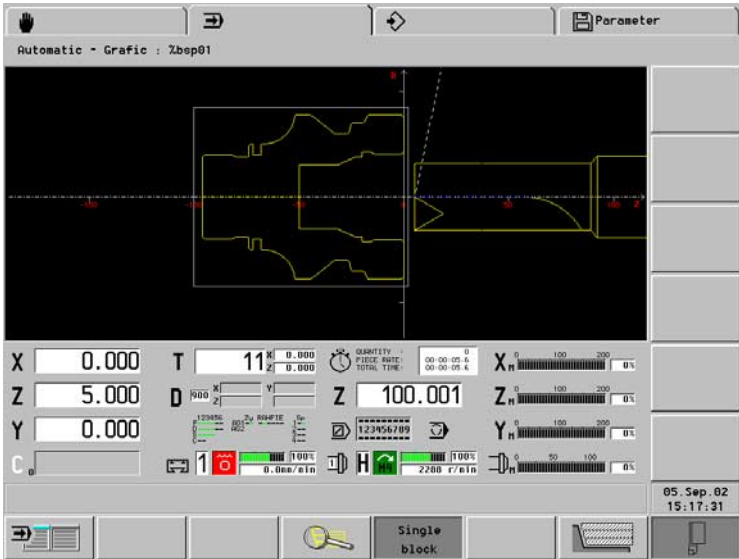
Standard: The system displays the complete path of traverse block by block

Motion: Depicts the metal removal in synchronism to the machining sequence.

Prerequisites:

- Workpiece blank is programmed
- Activate the "motion" function before starting the NC program.
- If an NC program is repeated (M99), the "Motion" function becomes active with the next program run.

Continued ►



Soft keys



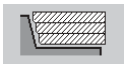
Return to block display



Activate the zoom function



Set to Single block mode



Depiction of traverse paths: **Line** or **(cutting) trace**




Tool depiction: **Point of light** or **tool**



- "Motion" appears only for lathes with one slide.
- If no blank part was programmed, the standard blank form (control parameter 23) is assumed.

Enlarging, reducing, selecting a section for enlargement

 When you call the zoom function, a red frame appears with which you can select the detail you wish to isolate.

Detail:

- Enlarge: "Page forward"
- Reduce: "Page back"
- Shift: Cursor keys


Zoom settings by touch pad

Prerequisite: Simulation in "stop condition"

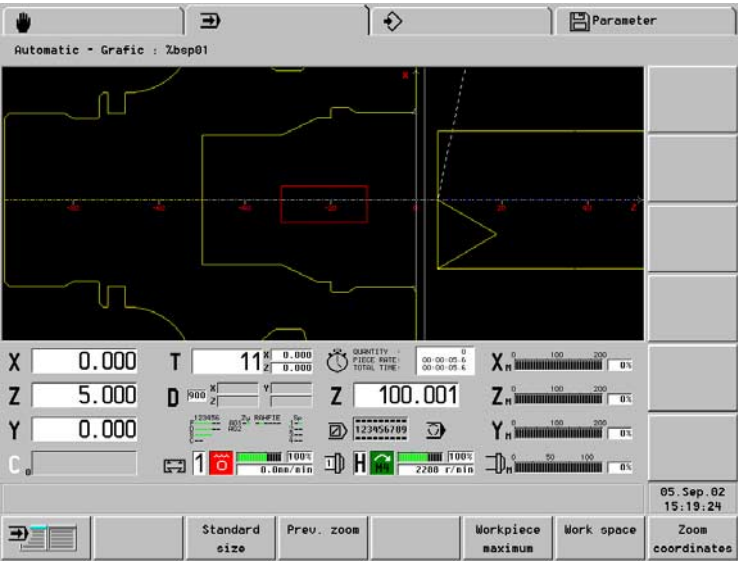
- Position the cursor to one corner of the section
- While holding the left mouse key, pull the cursor to the opposite corner of the section

Right mouse key: Return to standard size




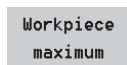
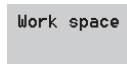

Standard settings: See soft-key table

 Exit the zoom function

After having enlarged a detail to a great extent, select "Workpiece maximum" or "Work space," and then isolate a new detail.



Soft keys

	Return to block display
	Cancels the zoom settings last used and displays the last standard setting ("Workpiece maximum" or "Work space").
	Switches back to the last zoom/setting used. You can select "Previous zoom" more than once.
	Shows the workpiece in the largest possible magnification
	Shows the working space including the tool change position.
	In the "Coordinate system" dialog box, you specify the dimensions of the simulation window and the position of the workpiece zero point.

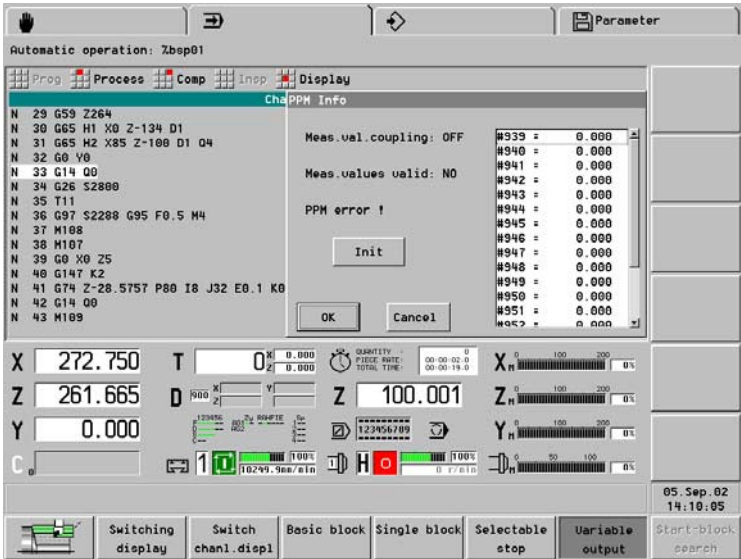
3.5.9 Post-Process Measuring Status Display

Selection: Menu item "Display – PPM Status" (automatic mode)

The "PPM Info" dialog box contains information on the status of the measured values and displays the transferred results:

- "Measured-value coupling" (control parameter 10)
 - Off: Measuring results are immediately transferred. Previous values are overwritten.
 - On: Measuring results are not transferred until the previous results have been processed.
- Measured values valid: Status of the measured values (after the measured values have been transferred with G915, the status "No" is displayed).
- #939: Total result of last measurement.
- #940..956: Measuring results last transferred by the measuring function.

If you select "Init," the post-process measuring function is re-initialized and all measured values are deleted.



The post-process measuring function stores the measured values received in the clipboard. The "PPM Info" dialog box displays in #939..956 the values contained in the clipboard - not the variables.

3.6 Machine Display

The machine display of the CNC PILOT can be configured. Per slide, you can configure up to 6 displays in Manual mode and Automatic mode.

Switching display

Switches to the "next configured display"

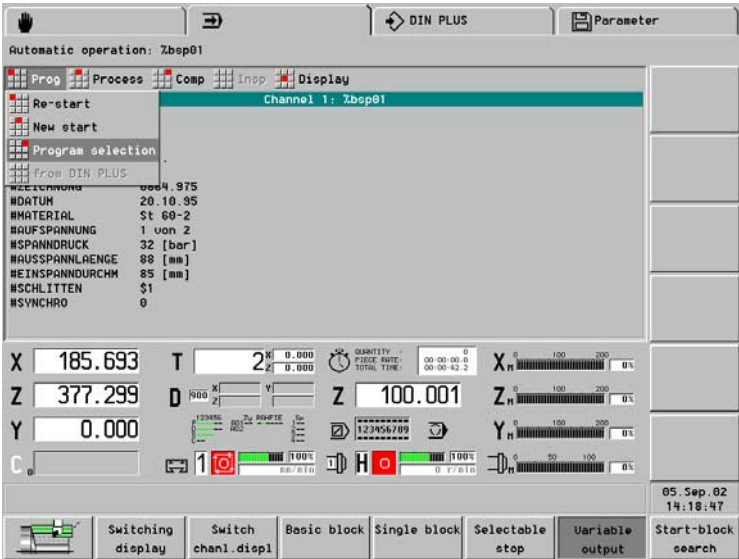
To switch to the display of the following slide, press the Change slide key. With the spindle change key you can display the next spindle.

The "display elements" table explains the standard display fields. For more display fields, see "7.3 Control Parameters"



You can set the values of the **Position display** in "Display setting" (machine parameter 17):

- 0: Actual values
- 1: Lag
- 2: Distance to go
- 3: Distance between tool tip and position of slide
- 4: Slide position
- 5: Distance between reference cams and zero pulse
- 6: Nominal position value
- 7: Distance between tool tip and slide position
- 8: IPO nominal position



Display elements

Position display (actual value display)



Distance from tool point to tool zero point

- Empty box: Reference mark in this axis not yet traversed
- White axis letter No enabling

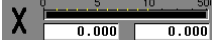
Position display (actual value display) C



Position of the C axis.

- "Index": Indicates the C axis „0/1“
- Empty box: C axis is not active
- White axis letter: No enabling

Distance-to-go display



Distance remaining in the current traverse command

- Bar graphic: Distance to go in millimeters
- Box at lower left: Actual position
- Box at lower right: Distance to go

T display – without tool life monitoring



- T number of active tool
- Tool compensation values

T display – with tool life monitoring

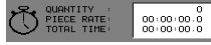


- T number of the active tool
- Tool life data

Continued ▶

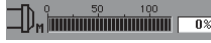
Display elements (continued)

Quantity of workpieces/time per workpiece



- Number of finished workpieces in this batch
- Machining time of current workpiece
- Entire production time of this batch

Load display



Load of the spindle motors/axis drives with regard to rated torque

D display – additive compensation



- Number of the active compensation
- Compensation values

Slide display



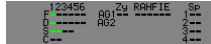
- White symbol: No enabling
- Number: Selected slides
- Cycle state: see table
- Bar diagram: Feed rate override "in %"
- Upper box: Feed rate override
- Lower box: Current feed rate – with stationary spindle: Nominal feed rate (gray print)
- Slide number in blue background: Rear side machining active

Spindle display



- White symbol: No enabling
- Number in spindle symbol: Gear range
- "H"/number: Selected spindle
- Spindle status: See table
- Bar diagram: Spindle speed override "in %"
- Upper box: Spindle speed override
- Lower box: Current speed – with position control (M19): Spindle position – with stationary spindle: Nominal speed (gray print)

Overview of enabled elements



Shows the enabling status of up to 6 NC channels, 4 spindles, and 2 C-axes. Enabled elements are marked in green.

- Display group at left: Enabled elements
F=feed rate; D=data; S=spindle; C=C-axis.
1..6: Number of slides/ of spindle, of C axis
- Display group at center: Status
Zy – left dash: Cycle on/off.
Zy – right dash: Feed Stop.
R=traversing the reference marks; A=automatic mode;
H>manual control;
F=retracting (after traversing the limit switches);
I = Inspection mode; E = Setup switch;
- Display group at right: Spindle
Display for "direction of rotation left/right."
Both active: Positioning of spindle (M19).

Cycle status (slide display)

Automatic mode – Cycle Start



Automatic mode – Feed Stop



Automatic mode – Cycle Stop



Manual control



Inspection cycle



Machine in setup mode



Spindle status (spindle display)

Direction of spindle rotation M3



Direction of spindle rotation M4



Spindle stopped



Spindle position-controlled (M19)



C-axis is "active"



3.7 Load Monitoring

The load monitoring function of the CNC PILOT compares the current torque, or the values for work, with the values from a "reference run."

If "torque limit 1" or the "work limit" is exceeded, the CNC PILOT marks the tool as "worn out." If "torque limit 2" is exceeded, the CNC PILOT assumes tool breakage and stops machining (feed stop). Violations of limit values are reported as error messages.

The load monitoring identifies worn tool in the "tool diagnosis bits." If you are using the **tool life management** function, the CNC PILOT will manage the replacement of tools (see "4.2.4 Tool Programming"). You can also evaluate the "tool diagnosis bits" in the NC program.

The load monitoring function defines the **monitoring zones** and the drives to be monitored (G995) in the NC program. The torque limits of a monitoring zone depend on the maximum torque determined by the reference machining cycle.

The CNC PILOT checks the values for torque and work in each interpolator cycle and displays the values in a time reference grid of 20 ms. The limit values are calculated from the reference values and the limit factor (control parameter 8). You can later change the limit values in "Edit load parameters."



- Make sure that the conditions for reference machining comply with those for production (feed-rate/speed override, tool quality, etc.).
- Up to four components are monitored per monitoring zone.
- Using "G996 Type of load monitoring," you can control the hiding the rapid traverses paths and the monitoring of torque and/or work.
- The graphic and numeric displays are relative to the rated torque values.

3.7.1 Reference Machining

The reference machining cycle (registration of nominal values) determines the **reference values** for the maximal permissible torque and work of each monitoring zone.

CNC PILOT executes a reference machining cycle if:

- The parameters for monitoring have not been entered.
- You press "Yes" in the "Reference machining" dialog box (after having selected the program).

Selection: "Display – Load monitoring – Load monitoring display" (Automatic mode).

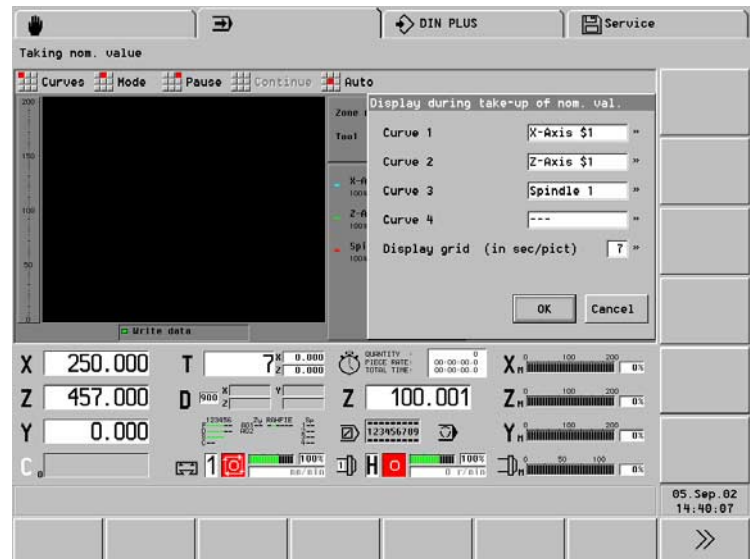
"Taking nominal values" submenu

■ "Curves" menu item

Assign the input fields "curves 1...4" to the drives. The value in "Display grid" influences the accuracy and velocity of the graphical display. A small value increases the accuracy of the display (values: 4, 9, 19, 39 seconds per image).

■ "Mode" menu group

- **Line graphics:** Display torque values over the time axis

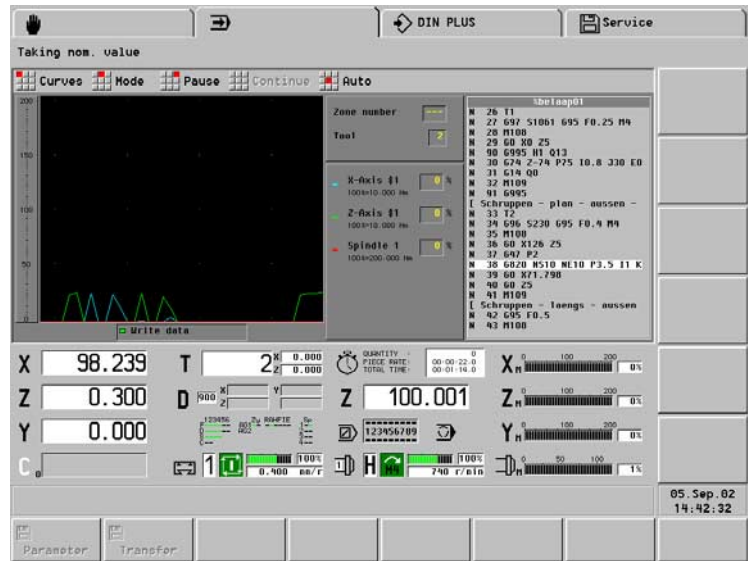


Continued ►

- **Bar graphic:** Graphically display torque values and mark the peak values
- **Save/do not save measured values**
The measured values must be saved for later analysis of reference machining. Check the setting in the "Save data" ("Write data") display field.
- **Overwrite/Do not overwrite limit values**
If you wish to retain the limit values after a new reference machining cycle, select "Do not overwrite limit values."
- **Pause** stops the display
- **Continue** continues the display
- **Auto:** returns to the automatic menu

Additional information

- **Zone number:** Current monitoring zone.
Negative algebraic sign: Production is not monitored (example: skipping the paths of rapid traverse).
- **Tool:** Active Tool.
- **Selected drives:** The drives are listed and the current torques are displayed.
- **Block display**



Block display has no influence on reference machining.

3.7.2 Production Using Load Monitoring

If you wish to use the load monitoring function for your machining processes, you must activate it in the NC program (G996).

Display torque values and limit values:

"Display – Load monitoring – Display" (Automatic operating mode).

"Load monitoring display" submenu:

"Curves" menu item

Enter the required drives in the input fields "Curves 1 to 4".

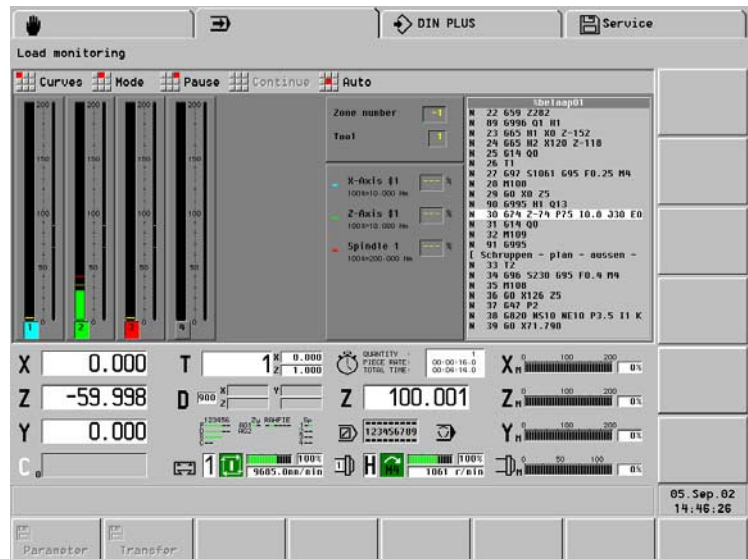
- **Line graphics:** One curve
- **Bar graphics:** Up to four curves

Display grid: See §3.7.1 Reference Machining"

"Mode" menu group

■ **Line graphic** Display the torque values over the time axis and limit values – limit values are gray: nonmonitored area (rapid traverse paths are hidden).

■ **Bar graphic** Displays current torque values, previous "work" and all limit values of the monitoring zone



- **Pause** Stops the display
- **Continue** Resumes the display
- **Auto** Returns to the automatic menu

3.7.3 Editing LimitValues

The function for editing the load parameters allows you to analyze reference machining cycles and optimize limit values.

The CNC PILOT displays the program name of the loaded monitoring parameters in the header.

Selection: "Display – Load monitoring – Edit"
(Automatic mode).

"Load parameter editor" submenu

- **"Act. load"** (Load current file) menu item: Monitoring parameters of the active NC program.
- **"Load"** menu item: Monitoring parameters you have selected.
- **"Edit"** menu item: Display and edit the limit values.
- **"Delete reference values"** menu item: Delete the load parameters of the NC program.
- **Auto** Returns to the automatic menu

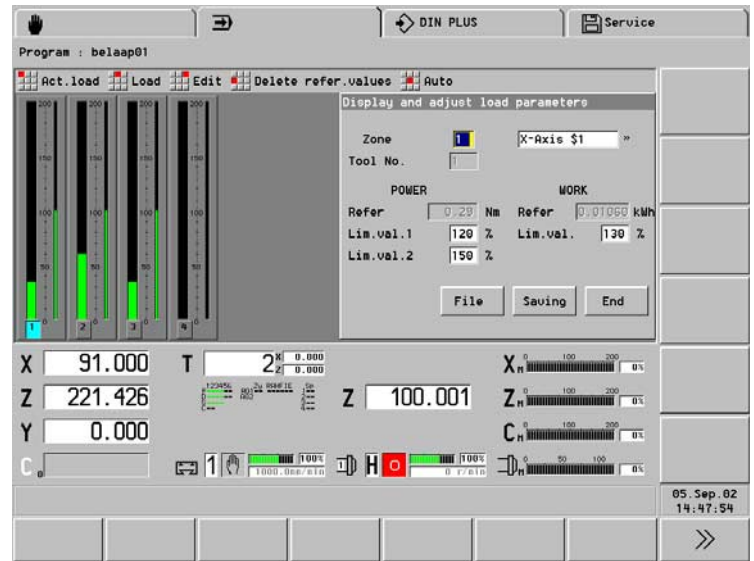
Editing the load parameters

The "Display and adjust load parameters" dialog box displays the parameters of **one** component **of one** monitoring zone, which can then be edited.

The bar graphic shows all components of the monitoring zone (the larger bar displays the values for performance; the smaller bar displays the values for work). The selected component is highlighted. Enter the monitoring zone and select the component. The CNC PILOT displays the reference values. The limit values for performance and work, which are displayed, can be edited. The tool (T number) is displayed for information.

Buttons of the dialog box:

- **Saving:** Store the limit values of the component in the specified zone.
- **End (or ESC key):** Exit the dialog box.
- **File:** Switch to "Line graphics." Precondition: The values measured during the reference machining cycle have been stored.



3.7.4 Analyzing Reference Machining

The torque and the limit values of the selected component are shown "over time." Limit values "gray": nonmonitored area (hiding rapid traverse paths).

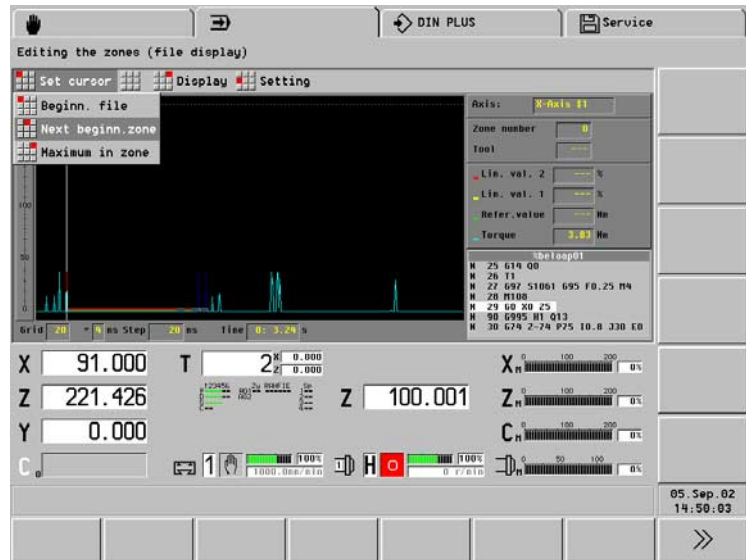
The CNC PILOT also displays the values of the cursor position.

Selection: "File" button ("Display and set load parameters" dialog box).

"Analyzer (file display)" submenu:

- **"Set cursor" pull-down menu;** using the right/left arrow key, position the cursor at the:
 - **Start of file**
 - **Start of the next zone**
 - **Zone maximum**
- **"Display" menu item:** Select the component from the "Display file" dialog box.
- **"Setting – Zoom" menu item:** Set the zoom factor. (Small values increase the accuracy of the display and reduce the step size of the cursor.) The settings defined for the grid, the time reference grid of the measured value registration and the cursor position are shown in the line below the graphic display (relative to the start of reference machining). Time "0:00.00 sec" = Start of reference machining cycle.

Esc Switch back to "Edit load parameters"



3.7.5 Machining Using Load Monitoring

It is advisable to use the load-monitoring function when - due to a dull cutting edge - a machining process requires a considerably higher torque than with a new cutting edge. As a rule, drives that are subjected to considerable loads should be monitored - usually the main spindle.

Due to the relatively small torque variations, it is difficult to monitor machining operations with small cutting depths.

A decrease in torque cannot be identified.

Defining the monitoring zones: The reference values for torque depend on the maximum torque values of the zone. Therefore, lower torque values cannot always be monitored.

Facing with constant cutting speed: The spindle is monitored as long as the acceleration is 15% of the mean value from maximum acceleration and maximum braking deceleration (machine parameters 811, ...). Since acceleration increases as a result of the increase in rotational speed, the CNC PILOT usually only monitors the period after the first cut.

Experimental values for the machining of steel

- For longitudinal turning, ensure that the cutting depth is greater than 1 mm.
- For recessing, ensure that the cutting depth is greater than 1 mm.
- For hole drilling, ensure that the diameter drilled is between 6 and 10 mm.

3.7.6 Load Monitoring Parameters

Machine parameters for load monitoring (spindle: 809, 859, ...; C-axis: 1010, 1060; linear axes: 1110, 1160, ...)

- **Start time for monitoring** [0 to 1000 ms] is calculated when rapid traverse movements have been skipped:
 - **Spindles:** A limit value is calculated from the acceleration and brake ramps. As long as nominal acceleration exceeds the limit value, the monitoring function is deactivated. If nominal acceleration drops below the limit value, the monitoring function is delayed by the "start time for monitoring."
 - **Linear axes and C-axis:** After rapid traverse has changed to feed rate, the monitoring function is delayed by the "start time for monitoring."
- **Number of measured values to be averaged [1..50]**
The mean value reduces the sensitivity to short peaks.
- **Maximum torque** of the drive [Nmm]
- **Delay in reaction P1, P2 [0 to 1000 ms]:** The CNC PILOT indicates that torque limit 1/2 has been exceeded after the time "P1/P2" has passed.

Control parameter 8 "Load monitoring settings"

- **Factor for torque limit value 1, 2**
- **Factor for work limit value**
Limit value = reference value * factor for limit value
- **Minimum torque [% of rated torque]:** Reference values below this value are raised to this minimum torque value. This prevents that limit values are exceeded as a result of minor differences in torque.
- **maximum file size [KB]:** If the data exceed the "maximum file size," the "oldest measured values" are overwritten. Approximate value: For one component per minute of program run time approximately 12 KB.

Control parameter 15 "bit codes for load monitoring":

Assigns the bits number used in G995 to the drives (logical axes).



4

DIN PLUS

4.1 DIN Programming

4.1.1 Introduction

The CNC PILOT supports conventional DIN programming and DIN PLUS programming.

Conventional DIN programming

You program the basic contour with line segments, circular arcs and simple turning cycles. For conventional DIN programming, the “simple tool description” is sufficient (see section “4.4.2Turret”).

DIN PLUS – Programming

The geometrical description of the workpiece and the machining process are separated. You first program the geometry of the blank and finished part. Then you machine the workpiece, using contour-related turning cycles. The **contour follow-up** function can be activated for each machining step, including individual paths of traverse and simple turning cycles. The CNC PILOT optimizes the machining process as well as the paths for approach and departure (no noncutting passes).

Depending on the type and complexity of your machining task, you can use simple DIN programming or DIN PLUS programming.

NC program sections

The CNC PILOT supports the division of the NC program into individual program sections. Sections containing set-up information and organizational data are included.

NC program sections:

- Program head (organizational data and setup information)
- Tool list (turret table)
- Chucking-equipment table
- Definition of blank
- Definition of finished part
- Machining of workpiece

Parallel operation

While you are editing and testing programs, your machine can execute **another** NC program.

Example: Structured DIN PLUS program

PROGRAMMKOPF [PROGRAM HEAD]

#MATERIAL	St 60-2
#EINSPANNDURCHM [CLAMPING DIAMETER]	120
#AUSSPANNLAENGE [CLAMPING LENGTH]	106
#SPANNDRUCK [CLAMPING PRESSURE]	20
#SCHLITTEN [SLIDE]	\$1
#SYNCHRO	0

REVOLVER 1 [TURRET]

T1 ID"342-300.1"
T2 ID"111-80-080.1"
T3 ID"112-16-080.1"
T4 ID"121-55-040.1"
T5 ID"122-20-040.1"
T6 ID"151-600.2"

SPANNMITTEL [CHUCKING EQUIPMENT] [zero offset Z282]

H1 ID"KH250"
H2 ID"KBA250-77" Q4.

ROHTEIL [BLANK]

N1 G20 X120 Z120 K2

FERTIGTEIL [FINISHED PART]

N2 G0 X60 Z-115
N3 G1 Z-105
...

BEARBEITUNG [MACHINING]

N22 G59 Z282
N23 G65 H1 X0 Z-152
N24 G65 H2 X120 Z-118
N25 G14 Q0
[Predrilling-30mm-Outside-Centric-Front face]
N26T1
N27 G97 S1061 G95 F0.25 M4
...
ENDE [END]

4.1.2 DIN PLUS Screen

- 1 Menu bar
- 2 Display of loaded NC programs. The selected program is marked.
- 3 Full, double or triple editing window. The selected window is marked.
- 4 Contour display (or machine display)
- 5 Soft keys

Parallel editing

You can edit up to eight NC program/subprograms in parallel. The CNC PILOT displays NC programs as desired in either a full, double, or triple window.

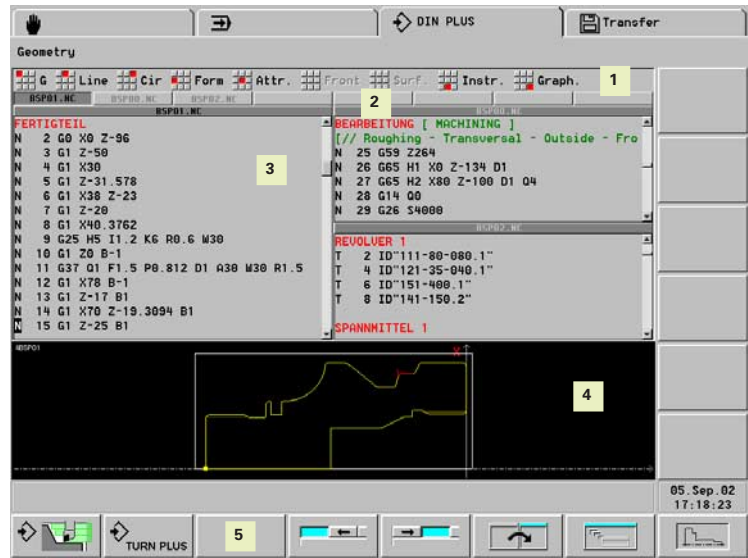
Main menus and submenus

The functions of the DIN PLUS editor are contained in the main menu and various submenus. The submenus can be called by

- Selecting the desired menu items
- Positioning the cursor in the program section

Soft keys

Soft keys are available for fast switching to "neighboring operating modes" for changing the editing window and for activating the graphic.



Soft keys



Change to the simulation operating mode



Change to theTURN PLUS mode



Switch the NC program



Switch the NC program



Switch the editing window



Select full-size window (one editing window)



Select double or triple window



Activate the graphics

4.1.3 Linear and Rotary Axes

Principle axes: Coordinates of the X, Y and Z axes refer to the workpiece zero point. Any deviations from this rule will be indicated.



Note for **negative X-coordinates:**

- Not permitted for contour definition.
- Not permitted for turning cycles.
- Contour regeneration is interrupted.
- The direction of rotation of arcs (G2/G3, G12/G13) must be adjusted manually.
- The position for tooth and cutter-radius compensation (G41/G42) must be adjusted manually.

C axis: Angle data are with respect to the zero point of the C axis. (Precondition: The C axis has been configured as a principal axis.)

For C-axis contours and C-axis operations, the following applies:

- Positions on the front/rear face are entered in Cartesian coordinates (XK, YK), or polar coordinates (X, C).
- Positions on the lateral surface are entered in polar coordinates (Z, C). Instead of C, the "linear value CY" is used ("unrolled" reference diameter).

Secondary axes (auxiliary axes): In addition to the principle axes, the CNC PILOT supports:

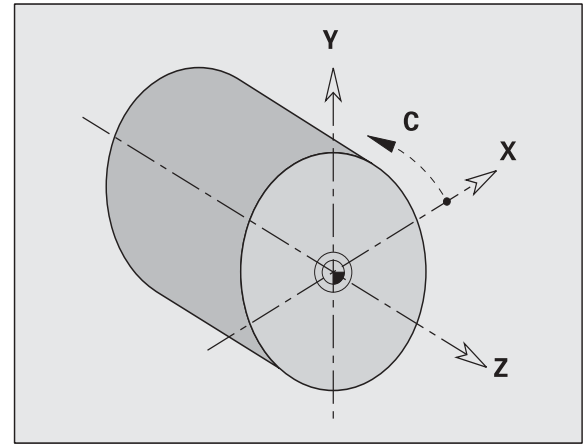
- U: Linear axis in X direction
- V: Linear axis in Y direction
- W: Linear axis in Z direction
- A: Rotary axis around X
- B: Rotary axis around Y
- C: Rotary axis around Z

The auxiliary axes are only programmed in the MACHINING section, using the functions G0 to G3, G12, G13, G30, G62 and G701. Circular interpolation is only possible in the principal axes.

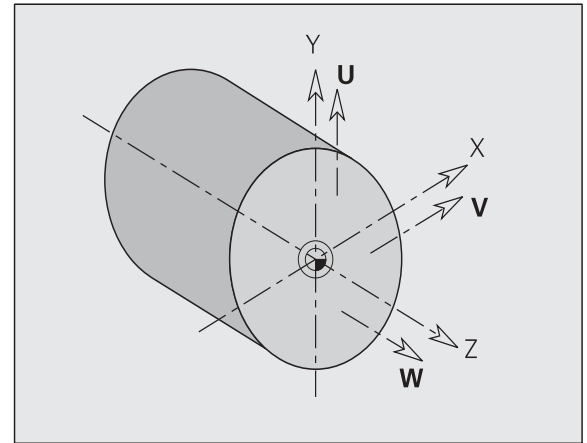
Rotary axes (auxiliary axes) are programmed in the MACHINING section, using G15.



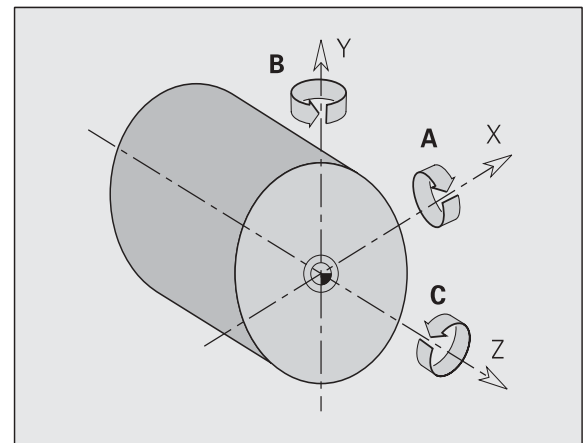
- The DIN editor respects only address letters of the configured axes.
- The behavior of the **rotary axis C** depends on whether it is configured as principle or secondary. The "C axis functions" G100..G113 apply for the principle axis C.



Principal axes



Linear axes as secondary axes



Rotary axes as secondary axes

4.1.4 Units of Measurement

You can use the metric or inch system for writing NC programs. The unit of measure is defined in the "Unit" box (see "4.4.1 Program Head"). After the unit of measure has been defined, it cannot be edited. Units of measure used: See "1.4 Fundamentals."

4.1.5 Elements of the DIN Program

A DIN program consists of the following elements:

- Program number
- Program section codes
- NC blocks
- Commands for structuring the program
- Comment blocks

The **program number** begins with "%" followed by up to 8 characters (numbers, upper case letters or underscore; no mutated vowels or "ß") and the extension "nc" for main programs or "ncs" for subprograms. The first character must be a number or a letter.

Program section code: When you create a new DIN program, certain program section codes are already entered. You can add new codes or delete existing ones, depending on your program requirements. A DIN program must contain at least the MACHINING and END section codes.

NC blocks start with the letter "N" followed by a block number (with up to four digits). The block numbers do not affect the sequence in which the program blocks are executed. They are only intended for identifying the individual blocks.

The NC blocks of the PROGRAM HEAD, TURRET and CHUCKING EQUIPMENT sections are not included in the "block number organization" of the DIN editor.

An NC block contains **NC commands** such as positioning, switching or organizational commands. Traversing and switching commands begin with "G" or "M" followed by a number (G1, G2, G81, M3, M30, ...) and the address parameters. Organizational commands consist of "key words" (WHILE, RETURN, etc.), or of a combination of letters/numbers.

You can also program NC blocks containing only variable calculations.

You can program various NC commands in an NC block provided that they do not contain the same address letters and do not have opposing functionalities.

Continued ►

Examples

- Permissible combination:
N10 G1 X100 Z2 M8
- Impermissible combination:
N10 G1 X100 Z2 G2 X100 Z2 R30 (same address letters used more than once)
or
N10 M3 M4 – opposing functionality

NC address parameters

Address parameters consist of 1 or 2 letter(s) followed by a

- A value
- A mathematical expression
- A "?" (simplified geometry programming VGP)
- An "i" to designate incremental address parameters (examples: Xi..., Ci..., XKi..., YKi..., etc.)
- A **# variable** (calculated during NC program interpretation)
- A **V variable** (calculated during run time)

Examples:

- X20 (absolute dimension)
- Zi-35.675 (incremental dimension)
- X? (Simple geometry programming)
- X#12 (Programming of variables)
- X{V12+1} (Programming of variables)
- X(37+2)*SIN(30) (Mathematical term)

Program branches and repeats

- You can use program jumps, repeats and subprograms to structure a program. Example: Machining the beginning/end of a bar etc.
- **Skip level:** Influences the execution of individual NC blocks
- **Slide code:** you can assign the NC blocks to the indicated slides provided that your lathe is equipped with more than one slide.

Input and output

With "input" the machine operator can influence the flow of the NC program. Using "output" functions, you can communicate with the machinist. Example: The machinist is required to check measuring points and update compensation values.

Comments

These are enclosed in parentheses "[...]" They are located at the end of an NC block or in a separate NC block.

4.2 Programming Notes

4.2.1 Parallel Editing

The CNC PILOT

- runs up to eight NC program/subprograms in parallel
- provides up to three editing windows

Editing window

Double or triple window: Selected in "Config – Window – ..." (main menu).

Load the desired NC program.

Load NC program in the next free window:

- ▶ Select "Prog – Load – Main program/Subprogram"

Load NC program in selected window:

- ▶ Select and activate free editing window
- ▶ Select "Prog – Load – Main program/Subprogram"

Switching between NC programs and windows

- By soft key: see table
- By touch pad:
 - To switch NC programs: Click the NC program in the program title bar
 - To switch editing windows: Click the desired window

Save the NC program

- "Prog – Save": Saves the NC program of the active window. The NC program stays in the editing window – you can continue editing it.
- "Prog – Save as": Saves the NC program of the active window under a new program name. In the "Saving NC program" dialog box you specify whether the editing window is closed.
- „Prog – Save all": Saves the NC programs of all active windows. The NC programs remain in the editing windows – you can continue editing them.

4.2.2 Address Parameters

You can use absolute or incremental coordinates for programming. If no entry is made for X, Y, Z, XK, YK, C, the coordinates of the block previously executed will be retained (modal).

CNC PILOT calculates missing coordinates in the principal axes X, Y or Z if you program "?" (simplified geometry programming).

The machining functions G0, G1, G2, G3, G12 and G13 are modal. This means that the CNC PILOT uses the previous G command if the address parameters X, Y, Z, I or K in the following block have been programmed without a G function provided that the address parameters have been programmed as absolute values.

"Switch windows" soft key



Switch the NC program



Switch the NC program



Switch the editing window



Select full-size window (one editing window)



Select double or triple window

Continued ►

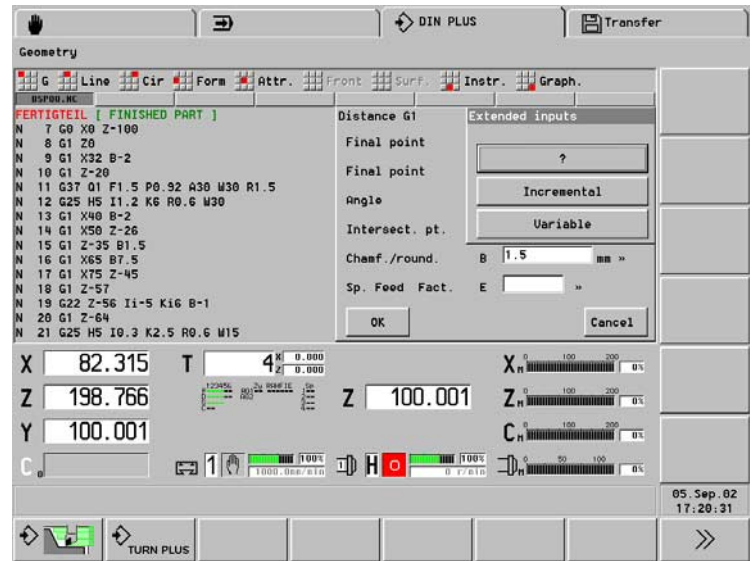
The CNC PILOT supports the use of variables and mathematical expressions as address parameters.

Editing address parameters

- Call the dialog box.
- Place the cursor in the required input box.
 - Enter/edit values, or
 - CONTINUE soft key: The "Extended input" dialog box appears
 - Program "?" (simplified geometry programming).
 - Switch from "Incremental" to "Absolute," or vice versa.
 - Activate the input of variables.



CNC PILOT only shows the "Extended inputs" permitted in the respective input box.



4.2.3 Contour Programming

The "contour follow-up" function and contour-turning cycles require the previous description of the blank and finished part. For milling and drilling with the C or Y axis, contour definition is a precondition if you wish to use fixed cycles.

Remember with **Contours for turning**:

- Describe a continuous contour.
- The direction of the contour description is independent of the direction of machining.
- CNC PILOT closes open contours paraxially.
- Contour descriptions must not extend beyond the turning center.
- The contour of the finished part must lie within the contour of the blank part.
- When machining bars, only define the required section as blank.
- Contour definitions are valid for the complete NC program, even if the workpiece is rechucked for machining the rear face.
- In the fixed cycles you program "reference values" referenced to the contour description.

Continued ►

To describe blank parts, use

- G20 "Blank part macro" for standard parts (cylinder, hollow cylinder).
- G21 "Cast-part macro" for blank-part contours based on finished-part contours.
- Individual contour elements (such as are used for finished-part contours) if use of G20 or G21 is not possible.

To describe finished parts, use individual contour elements. The contour elements or the complete contour can be assigned attributes which are accounted for during the machining of the workpiece (example: roughness, allowances, etc.).

For intermediate machining steps, define **auxiliary contours**. Auxiliary contours are programmed in the same way as finished-part descriptions. You can program one contour definition per AUXILIARY CONTOUR. The number of auxiliary contours in a program is not limited.

Contours for machining with the C/Y axis

Contours that are milled or drilled are programmed within the FINISHED PART section. The machining planes are defined as FRONT, FRONT_Y, SURFACE, SURFACE_Y, etc. You can repeatedly use the section codes, or program various contours within one section code.

Up to four contour per NC program

The CNC PILOT support up to four contour groups (workpiece blank and finished part) in one NC program.

The code CONTOUR introduces the description of a contour group. Parameters on zero point shift and the coordinate system define the position of the contour in the working space. A G99 in the machining section assigns the machining to a contour.

Contour generation during simulation:

You can save contours generated in the simulation and insert it in the NC program. Example: You describe the workpiece blank and finished part, and simulate the machining of the first setup. Then you save the contour. You define a shift of the workpiece zero point and/or a mirror image. The simulation saves the "generated contour" as the workpiece blank and the originally defined finished part contour - taking the zero point shift and mirroring into account.

In DIN PLUS, you insert into the program the workpiece blank and finished part contour that you generated during simulation (block menu – "Insert contour").

Contour follow-up

CNC PILOT takes the blank part as a basis and accounts for each cut and each cycle of the turning operation when following up the contour. Thus you can inspect the current contour of the workpiece during each machining stage. With the "contour follow-up" function, the CNC PILOT optimizes the paths for approach and departure and avoids noncutting passes.

Continued ►

The contour follow-up function can also be used for auxiliary contours.

Preconditions for contour follow-up:

- Definition of blank
- Proper description of tools ("simple definition of tools" is not sufficient)

The contour follow-up function can be used only for turning contours; **it cannot be used for** contours with the C or Y axis.

Contour simulation

During editing CNC PILOT displays programmed **contours** in up to two **graphic windows**.

- Selection of the graphic window: "Graphic – Window" menu item
- Back to machine display: "Graphic – Graphic OFF" menu item



Activate graphic window or update the contour

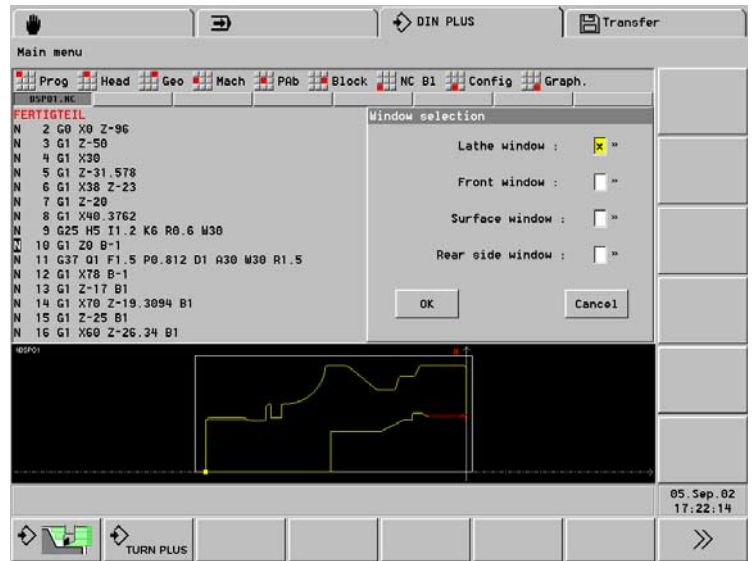
Note:

- The starting point of the turning contour is marked by a "small box."
- When the cursor is located on a block of the "BLANK or FINISHED PART" section the corresponding contour element is highlighted in red and the direction of contour definition is indicated.
- When programming fixed cycles, you can use the displayed contour for establishing block references.
- The CNC PILOT starts at the pattern base when displaying contours on lateral surfaces (reference diameter for SURFACE).

4.2.4 Tool Programming

The designations of the tool pockets are fixed by the machine tool builder. Each tool holder has a unique **T number**.

In the "T command" (MACHINING section) you program the position of the tool holder, and therefore the position to which the tool carrier rotates. The CNC PILOT retrieves the assignment of the tools to the turret positions from the TURRET section, or the tool list (in case the T number is not defined in TURRET).



- Additions/changes on the contours are not considered until GRAPHIC is selected again.
- Unambiguous NC block numbers are a prerequisite for the "contour display"!

Continued ►

Multipoint tools

For tools with more than one point (multiple tools) the T number is followed by an

„.S“:

T number S S: Number of the cutting edge [0 to 4]
(0=main cutter - may be omitted)

In the TURRET section, you define only the main cutting edge.

When a cutting edge of a multiple tool has become dull, the tool life management function marks all cutting edges "worn-out."

Examples:

- T3 or T3.0 – Turret position 3; Main cutting edge
- T12.2 – position to which the turret rotates 12; cutting point 2

Replacement tools

If you wish to use the **Tool life monitoring** function, you must define an "tool interchange chain." As soon as a tool is worn out, the CNC PILOT interchanges a replacement tool. The CNC PILOT does not stop the program run until the last tool of the tool interchange chain is worn out.

In the TURRET section and the T commands, you program the first tool to be interchanged. The CNC PILOT inserts the replacement tool automatically.

When programming variables (access to tool compensation or tool diagnosis bits), you also address the first tool of the chain. The CNC PILOT automatically addresses the "active tool."

You can define replacement tools in "Setup" (see "3.3.4 Tool Life Management").

4.2.5 Fixed cycles

HEIDENHAIN recommends programming a fixed cycle in the following steps: (see: "4.18.1 Programming a Fixed Cycle"):

- Insert tool.
- Define the cutting data.
- Position the tool in front of the working area.
- Define the safety clearance.
- Call a cycle.
- Retract the tool.
- Approach the tool change position.



Danger of collision!

If cycle-programming steps are omitted when a program is optimized:

- A special feed rate remains in effect up to the next feed-rate command (example: finishing feed for recessing cycles).
- Several cycles return diagonally to the starting point if you use the standard programming (example: roughing cycles).

4.2.6 NC Subprograms

Subprograms are used to program the contour or the machining process.

In the subprograms, transfer parameters are available as variables. You can fix the designation of the transfer parameter (see "4.16 Subprograms").

In every subprogram, the variables #256 to #285 are available for internal calculations.

Subprograms can be nested up to six times. This means that a subprogram calls in a further subprogram, etc.

If a subprogram is to be executed repeatedly, enter the number of times the subprogram is to be repeated in the parameter Q.

The CNC PILOT distinguishes between **local** and **external subprograms**. Local subprograms and the NC main program are stored in the same file. Local subprograms can only be called in from their corresponding main programs. External subprograms are stored in separate NC files and can be called in from any NC main program or other NC subprograms.

Expert programs

The machine manufacturer usually provides subroutines, which are tailored to the machine configuration, for complex processes such as workpiece transfer for full-surface machining. (Example: workpiece transfer for full-surface machining). Refer to the machine manual.

4.2.7 Template Control

"Templates" are predefined NC code blocks integrated in the NC program. They reduce programming input and help standardize the program format.

Templates are defined by the machine tool builder. Your machine tool builder can tell you whether he offers templates and how they can be used.

4.2.8 NC Program Interpretation

For variable programming and user communication, keep in mind that the CNC PILOT interprets the complete NC program **before** it can be run (see "3.5 Automatic Mode of Operation").

The CNC PILOT differentiates between:

- **#-variables** are calculated during the interpretation of the NC program.
- **V-variables** are calculated at runtime, which means during the execution of an NC block.
- Input/output during **NC program interpretation**.
- Input/output during **NC program run**.

4.3 The DIN PLUS Editor

Select menu items

The submenus can be called by

- Selecting the desired menu items
- Positioning the cursor in the program section

Esc

From the submenu back to the main menu

When you call the menu items "**G**eometry," "**P**rocessing," "Turret assignment" or "Chucking equipment," the CNC PILOT jumps to the corresponding program section. -When you position the cursor in the BLANK, FINISHED PART or MACHINING section, the CNC PILOT switches to the corresponding submenu.

Creating NC blocks

The insertion of new NC blocks varies depending on the program section.

- After the "Editing program head" dialog box has been concluded, the CNC PILOT automatically creates the blocks of the program head (code "#").
- In the TURRET and CHUCKING EQUIPMENT sections, you can insert a new block by pressing the INS key.
- When you program a contour or a machining process, or within a subprogram, the CNC PILOT automatically creates new NC blocks. Alternately, you can add NC blocks by pressing the INS key.

The new NC block is inserted **below** the cursor position.

Deleting elements of an NC block

- ▶ Position the cursor on an element of the NC block (NC block number, G or M command, address parameter, etc.), or the section code.
- ▶ Press the DEL key. The element highlighted by the cursor **and** all the related elements are deleted. (Example: If the cursor is located on a G command, the address parameters are also deleted.)

Editing elements of an NC block

- ▶ Position the cursor on an element of the NC block (NC block number, G or M command, address parameter, etc.), or the section code.
- ▶ Press ENTER or double-click with the left mouse key. The CNC PILOT activates a dialog box which displays the block number, the number of the G or M function, or the address parameters of the G function, which can then be edited.

When you edit NC words (G, M, T), the CNC PILOT additionally activates a dialog box for editing the address parameters.

When editing section codes, you can only change the associated parameters (Example: Number of the turret).



Before deleting a complete NC block, CNC PILOT displays a confirmation request. Individual elements of an NC block including G or M functions are deleted immediately.

Continued ►

"Conversational" or "free" editing

You usually select the NC functions from the menus and edit the address parameters in dialog boxes. You can also select "Free input" ("NC BI" pull-down menu) and edit the NC program. For "free editing," the maximum length of a block is 128 characters per line.

Block references

When editing G commands related to the contour (MACHINING section), you can switch to contour simulation and select the block references from the contour displayed, using the arrow keys.

G commands

The G commands are divided into:

- **Geometry commands** for describing the blank and finished part.
You can use additional "auxiliary commands" (allowance, surface quality, etc.) to influence the machining process.
- **Machining commands** for the MACHINING section.



Some G functions are used for blank/finished-part definition and in the MACHINING section. When copying or shifting NC blocks, keep in mind that "geometry" functions are used only for describing a contour, while "machining" functions are used only in the MACHINING section.

4.3.1 Main Menu**"Prog" pull-down menu (NC program management):**

- **Load** – loads stored NC programs:
 - ▶ The CNC PILOT displays existing NC main programs or subprograms.
 - ▶ Select the NC program
- **New** – creates new NC main **programs** or subprograms:
 - ▶ Enter a program name.
 - ▶ Select main program or subprogram.
 - ▶ To activate the "Editing program head" window, select "Program head."
- **Close** – Closes the selected NC program without saving it
- **Save** – Saves the selected NC program – the program stays open for editing
- **Save as** – Saves the selected NC program under a given name
 - ▶ "Do not close/Close": Select whether to close the editing window or leave it open to continue editing the NC program
 - ▶ "Save as ...": Enter the program name
- **Save all** – Saves all loaded NC programs



When you exit "DIN PLUS" operating mode, the NC programs are saved automatically. The old version of the NC program is overwritten.

"Head" pull-down menu (NC program head):

- **Program head:** activates the "Editing program head" dialog box.
- **Turret assignment:** positions the cursor in the TURRET section.
- **Chucking equipment:** positions the cursor in the CHUCKING EQUIPMENT section.

Continued ►

"Geometry" pull-down menu (contour programming):

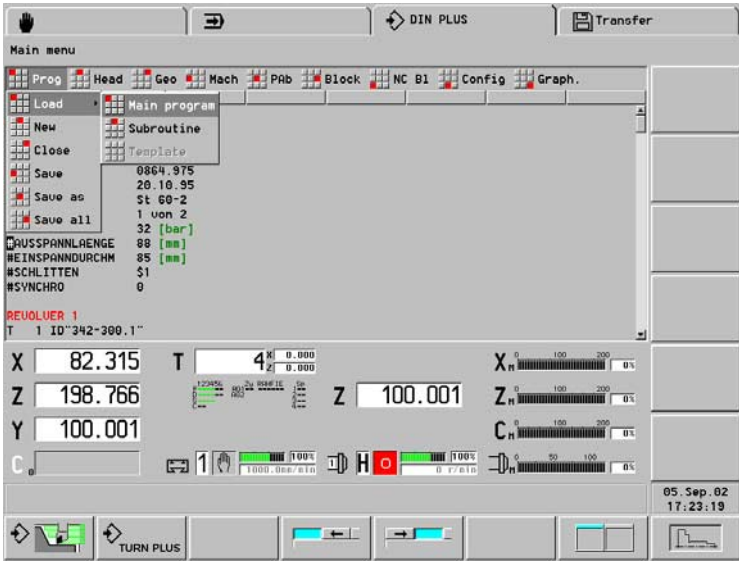
- **Blank – Chuck piece/bar G20:** creates an NC block in the BLANK section, switches to the "Geometry" menu and activates the "Chuck part cylinder/tube G20" dialog box.
- **Blank – Casting G21:** creates an NC block in the BLANK section, switches to the "Geometry" menu and activates the "Casting G21" dialog box.
- **Blank – Free contour:** positions the cursor in the BLANK section and switches to the Geometry menu.
- **Finished part:** positions the cursor in the FINISHED PART section and switches to the Geometry menu.

Single menu items

- **Programming:** switches to the Machining submenu and positions the cursor in the MACHINING section.
- **PAb (Program Section codes)** – inserts new section codes:
 - ▶ Select the desired section code and press ENTER.
 - ▶ The CNC PILOT inserts the section code at the correct position.
- **Block:** switches to the block-editing functions (see section "4.5.5 Block menu").

"NC BI" pull-down menu

- **Program beginning** Positions the cursor to the beginning of the program
- **Program end:** Positions the cursor to the program end
- **Search function - Search block**
 - ▶ Enter the block number.
 - ▶ The CNC PILOT positions the cursor on the block number provided that it exists.
- **Search function - Search word**
 - ▶ Enter the NC word to be searched for (G command, address parameter, etc.).
 - ▶ The cursor skips to the first NC block containing the word searched for. The CNC PILOT searches from the cursor position to the end of the program, then continues searching from the start of the program.
- **Increment** Numerical interval between NC blocks
The increment entered remains in effect for the active NC program only.



DIN PLUS main menu

	Prog (NC program management)
	Head: Edit the NC program head (program head, turret assignment, chucking equipment table)
	Geo: Program the contour of the blank and finished part (submenu "Geometry")
	Pro(gramming): Program the machining of the workpiece ("Machining" submenu)
	PAb: Insert program section codes
	Block: Switch to the block submenu containing functions for moving, copying or deleting NC blocks
	NC BI: Functions for block numbering, searching and "free" editing
	Configuration of the DIN PLUS screen display (with/without graphical display)
	Graphics: Select graphic simulation window, switch the contour simulation ON/OFF

Continued ►

■ **Block numbering:** The number "interval" is specified for the first NC block; for each following block the "interval" is added. Block references in contour-related G commands and subprogram calls are corrected automatically. The sequence of the NC blocks is retained.

■ **New: free input**

- ▶ Position the cursor.
- ▶ Select "New: free input."
- ▶ Enter NC block
- ▶ The new NC block is inserted below the cursor position.

■ **Modify: free input**

- ▶ Position the cursor on the NC block to be edited.
- ▶ Select "New: modified input."
- ▶ Edit the NC block

"Config(uration)" menu group:

■ **Aux. pict.:** Select whether you wish to display the help graphics.

■ **Window – Full-size window/Double window/Triple window:** Select the number of editing windows

■ **Font size – smaller/larger:** Change the font size within the editing window

■ **Font size – Adjust fonts:** Set the font size of the selected window in all editing windows

■ **Settings – Save:** Saves the current editor condition (window setting, all loaded NC programs)

■ **Settings – Load:** Loads the last save condition of the editor

■ **Settings – Auto-save on:** Saves the current editor condition when CNC PILOT is switched off

■ **Settings – Auto-save off:** No saving of the editor condition when the CNC PILOT is switched off

"Graphics" pull-down menu

■ **Graphic ON:** activates contour simulation.

■ **Graphic - OFF:** deactivates contour simulation and activates the machine display.

■ **Window** (selection of simulation window): Select a maximum of two windows. The contour simulation is activated by selecting "Graphic ON."

4.3.2 Geometry Menu

The "Geometry" submenu contains G functions and instructions for the BLANK and FINISHED PART sections.

Selecting a **G function**:

- The G number is known: Select "G" and enter the number.
- The G number is **not** known:
 - ▶ Select "G."
 - ▶ Press the CONTINUE soft key
 - ▶ Select the G function from the list of G numbers
- "G menu": To select the desired G function, use the pull-down menu.

"Instr(uctions)" pull-down menu:

- **DIN PLUS words** – calls selection list containing:
 - Instructions for structuring the program.
 - Instructions for input/output.
 - Section codes for contours with the C/Y axis.
- **Variables** – Enter a variable or mathematical expression
- **FRONT, SURFACE, REAR SIDE**
 - ▶ The dialog box for entering the contour position (reference plane/reference diameter) appears.
 - ▶ Enter the Z position/diameter
 - ▶ The CNC PILOT inserts the section code below the position of the cursor.
- **AUXILIARY CONTOUR (TEMPORARY)** – inserts the section code below the position of the cursor.
- **Comment line** – Enter a comment. The comment is inserted above the position of the cursor.

Single menu item:

- **Graphic** – Activates/deactivates contours in the graphic window.



Geometry submenu



G: Direct input of the G number / Calls the G list



Line: Activates the G1 Geo dialog box



Cir Clockwise arc, counterclockwise with incremental or absolute center dimensioning



Form Elements of the contour, subprogram call, reference plane for pocket/island



Attributes (auxiliary commands) for contour definition



Front: Basic elements, figures or patterns of the contour on the front or rear face (machining with the C axis)



Surface: Basic elements, figures or patterns on the lateral surface (machining with the C axis)



Instructions for structuring the program and for section codes



Graphics: Activate/update the contour in the graphic simulation windows

4.3.3 Machining Menu

The “Machining” submenu contains G and M functions as well as further functions for the MACHINING section.

Selecting a **G function**:

- The G number is known: Select “G” and enter the number.
- The G number is **not** known:
 - ▶ Select “G.”
 - ▶ Press the CONTINUE soft key
 - ▶ Select the G function from the list of G numbers
- “G menu”: To select the desired G function, use the pull-down menu.

Selecting an **M function**:

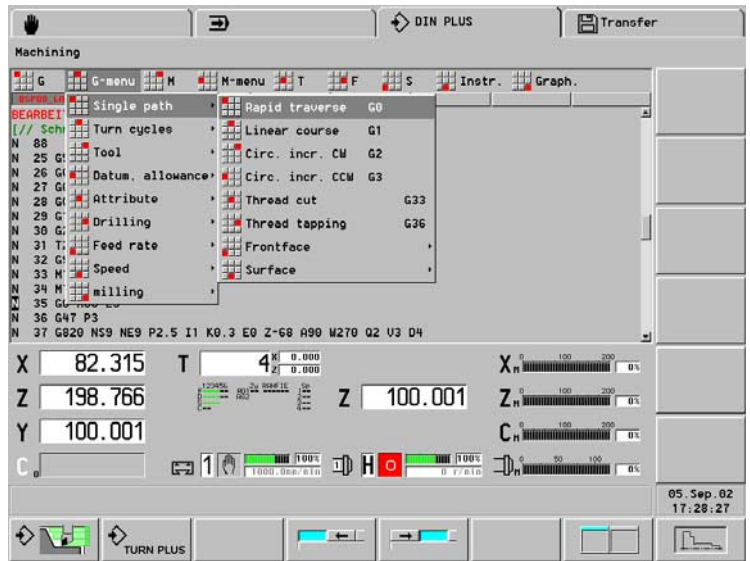
- The M number is known: Select “M” and enter the number.
- M menu: To select the M function, use the menu.

Single menu items

- **T** – Tool call
Program the T number (see “4.6.7Tools, Types of Compensation”). A list containing the tools indicated in the TURRET section is displayed.
- **F**- calls “G95 - “Feed per revolution.”
- **S**- calls “G96 – Cutting speed.”

“Instr(uctions)” pull-down menu:

- **DIN PLUS words** – calls selection list containing:
 - Instructions for structuring the program.
 - Instructions for input/output. n
- **Variables** – Enter a variable or mathematical expression
- **/ Skip level**
 - ▶ Enter skip level 1 to 9.
 - ▶ The CNC PILOT inserts the skip level before the NC block (example: /3 N 100 G...).
- **\$ Slide:**
 - ▶ Enter the “Slide number”; more than one slide number -one after the other- can be entered.
 - ▶ The DIN editor inserts the slide number before the NC block (example: \$1\$2 N 100 G...).



Machining submenu

	G: Enter number of G function/call the list of G functions
	G menu: Pull-down menus containing G functions appear
	M: Enter the M number
	M Menu: Opens pull-down menus containing M functions
	T: Tool call
	F: Call “G95 - “Feed per revolution”
	S: Calls “G96 – Cutting speed”
	Instructions for structuring the program
	Graphics: Activate/update the contour in the graphic simulation windows

Continued ►

- **L call external** – (see "4.16 Subprograms").
 - ▶ Select the subprogram and press RETURN
 - ▶ Enter the transfer parameters.
 - ▶ The CNC PILOT inserts the subprogram call.
- **L call internal** – (see "4.16 Subprograms").
 - ▶ Enter the name of the subprogram (number of the first block of the subprogram).
 - ▶ Enter the transfer parameters.
 - ▶ The CNC PILOT inserts the subprogram call.
- **Comment line**
 - ▶ Enter the comment – the comment is inserted above the cursor position.
- **Template selection** – Select from the available templates. Prerequisite: The machine manufacturer has defined templates
- **The working plan** – "collects" all comments that begin with "// ..." and places them before the MACHINING section. This gives you a summary of the functions in the NC program or subprogram.

Menu item:

- **Graphic** – Activates/deactivates contours in the graphic window.

4.3.4 Block Menu

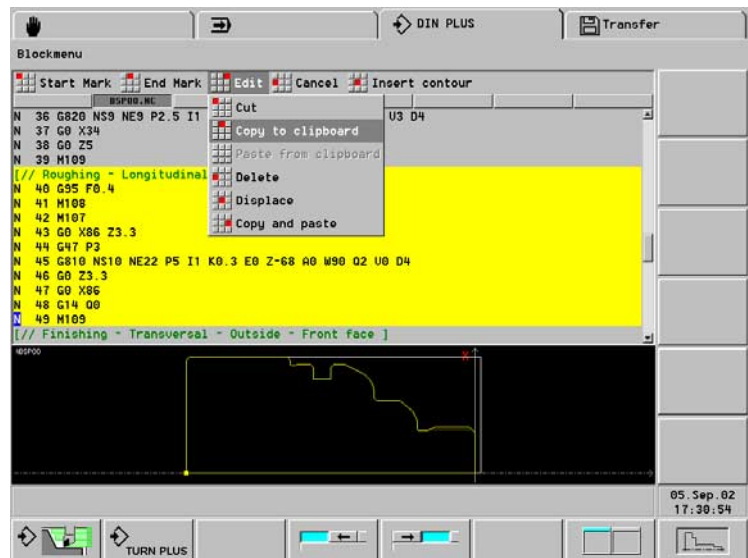
This function enables NC program sections (several successive NC blocks) to be moved, copied, deleted or exchanged between NC programs.

To define an NC block, highlight the first and last line of the block. Then select the desired function from the "Edit" menu.

In order to **exchange blocks between NC programs**, copy the block to the clipboard. Then read in the block from the clipboard. A block remains in the clipboard until it is overwritten by another block.

Single menu items

- **Start Mark:**
 - ▶ Position the cursor on the beginning of a block.
 - ▶ Press "Start mark."
- **End mark:**
 - ▶ Position the cursor on the end of a block.
 - ▶ Press "End mark."



Continued ►

"Edit" pull-down menu

- **Cut:**
 - Stores the highlighted block in the clipboard.
 - Deletes the block.
- **Copy to clipboard** – copies the highlighted block to the clipboard.
- **Paste from clipboard:**
 - ▶ Position the cursor where you wish to insert the block.
 - ▶ Select "Paste from clipboard."
 - ▶ The block is inserted at the position indicated by the cursor.
- **Delete** – deletes the highlighted block definitively (it is **not** stored in the clipboard).
- **Displace:**
 - ▶ Position the cursor where you wish to insert the block.
 - ▶ Select "Displace."
 - ▶ The highlighted block is moved from its initial position to the position indicated by the cursor.
- **Copy and paste:**
 - ▶ Position the cursor where you wish to insert the block.
 - ▶ Select "Copy and paste."
 - ▶ The block is inserted at/copied to the position indicated by the cursor.

Single menu items

- **Cancel** – all markings are canceled.
- **Insert contour** – inserts the most recent workpiece blank and finished part contour in the simulation below the cursor position

As an alternative of the block menu, you can use the usual **WINDOWS key combinations** for marking, deleting, shifting etc.:

- Marking by moving the cursor keys while holding the shift key.
- Ctrl + C: Copy the marked text to the clipboard
- Shift + Del(ete): Delete the marked text and save it in the clipboard
- Ctrl + V: Insert text from the clipboard at the cursor position
- Del(ete): Delete the marked text

4.4 Program Section Codes

A new DIN program is already provided with section codes. You can add new codes or delete existing ones, depending on your program requirements. A DIN program must contain at least the MACHINING and END section codes.

Overview of program section codes

PROGRAMMKOPF [PROGRAM HEAD]
TURRET
MAGAZINE
CHUCKING EQUIPMENT
KONTUR [CONTOUR]
ROHTEIL [BLANK]
FERTIGTEIL [FINISHED PART]
HILFSKONTUR [AUXILIARY CONTOUR]
BEARBEITUNG [MACHINING]
ENDE [END]
UNTERPROGRAMM [SUBPROGRAM]
RETURN

Machining with the C axis

FRONT
REAR SIDE
SURFACE

4.4.1 PROGRAMMKOPF [PROGRAM HEAD]

The PROGRAM HEAD comprises:

- **Slides:** NC program is run only on the given slides (input: „\$1, \$2, ...“) – no input: NC program is run for **every** slide.
- **Unit:** Unit of measure “metric/inches” – no input: The unit of measure defined in control parameter 1 is used
- The other codes contain **organizational information** and **set-up information** that do not influence the program run.

Information contained in the program head is preceded by “#” in the DIN program.



The “Unit” can be programmed only when a new program is being created (set under PROGRAM HEAD). It is not possible to post-edit this entry.

Definition of the variable display

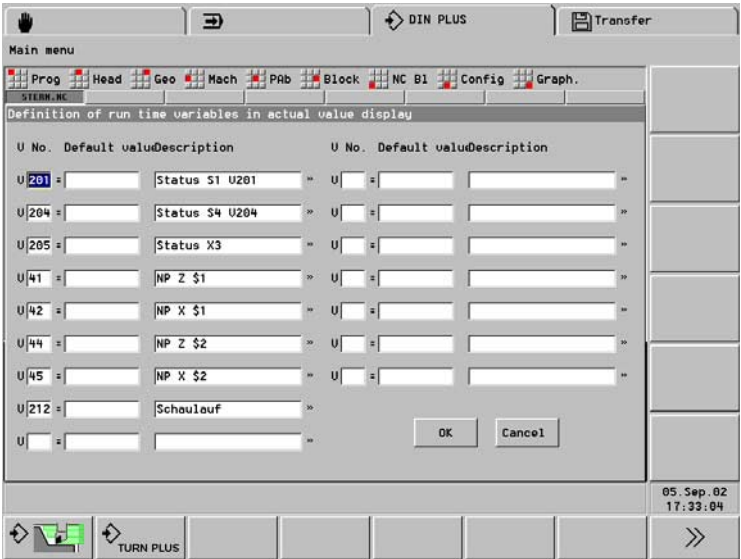
Call: "Variable display" button in the "Editing program head" dialog box

In the dialog box you define up to 16 V variables that control the program process. In automatic mode and in the simulation you define whether the variables are to be asked for during program run. As an alternative, the program version is run with the default values.

For each variable you define:

- Variable number
- Default value (initialization value)
- Description (text, with which this variable is asked for during program run)

The definition of the variable display is an alternative to programming with INPUTA/PRINTA commands.



4.4.2 TURRET

TURRET x (x: 1..6) defines the turret assignment of tool carrier x. You enter the ID number directly ("Tools" dialog box), or you take it from the tool database. You can access the tool database with the "Type list" or "ID list" soft key.

Alternately, you can define the tool parameters in the NC program.

Enter the tool data:

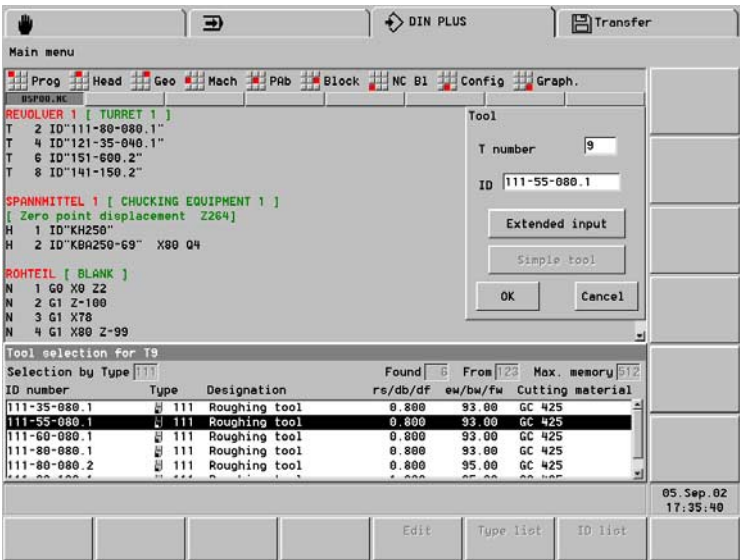
- ▶ „Select "Head – Turret assignment."
- ▶ Position the cursor in the TURRET section.
- ▶ Press the INS key.
- ▶ Edit the "Tool" dialog box.

Edit the tool data:

- ▶ Position the cursor.
- ▶ Press RETURN or double-click with the left mouse key
- ▶ Edit the "Tool" dialog box.

Parameters of the "Tool" dialog box:

- **T-number:** Position on the tool carrier.
- **ID (ID number):** Reference to the database – no input: Data saved in the database as "temporary tools."



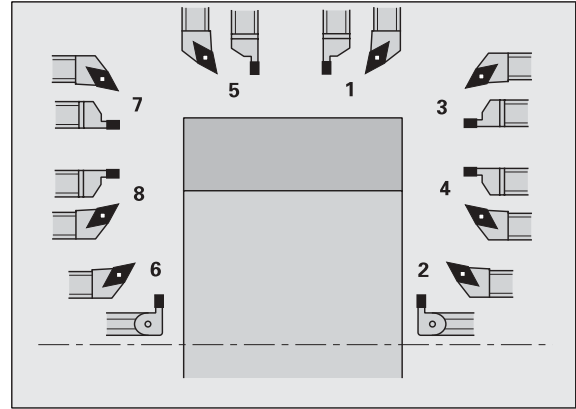
Access to the tool database by soft key

Edit	Edit the tool parameters
Type list	Entries in the tool database - sorted by tool type
ID list	Entries in the tool database - sorted by tool ID number

Continued ▶

- **Extended input:**
 - No constraints for the use of the tool.
 - Only the tool tip is shown in the simulation.
 - First you define the tool type, then you edit the tool parameters. The tool parameters are shown in the parameters of the first dialog box to the tool editor (see "8.1 Tool Database").
 - The data are saved in the database during program compilation only if you enter the ID number.
- **Simple tool:**
 - Only suitable for simple traverse paths and turning cycles (G0...G3, G12, G13; G81...G88).
 - No contour regeneration.
 - Tooth/cutter radius compensation.
 - Simple tools are not included in the database.
 - For the meaning of the parameters, refer to the table below.

Simple tools		
Dialog box	NC program	Meaning
Tool type	WT	Tool type and machining direction
X dimension (xe)	X	Setup dimension
Y dimension (ye)	Y	Setup dimension
Z dimension (ze)	Z	Setup dimension
Radius R (rs)	R	Cutting radius of turning tools
Cutting width B (sb)	B	Cutting width of recessing and button tools
Diameter I (df)	I	Milling or drilling diameter



- If you **do not program the TURRET**, the tools entered in the "tool list" are used (see "3.3.1 Setting Up the Tool List").
- The names "_SIM..." and "_AUTO..." are reserved for temporary tools (simple tools and tools without ID number). Tool entries are valid as long as the NC program is activated in the simulation or Automatic mode.

Example:TURRET --Table
REVOLVER 1 [TURRET]
T1 ID "342-300.1" [Tool from the database]
T2 WT1 X50 Z50 R0.2 B6 [simple tool description]
T3 WT122 X15 Z150 H0V4 R0.4 A93 C55 I9 K70 [extended tool description – not transferred to database]
T4 ID "Erw.1" WT112 X20 Z150 H2V4 R0.8 A95 C80 B9 K70 [extended tool description – with transfer to database]
...

4.4.3 CHUCKING EQUIPMENT

CHUCKING EQUIPMENT x (x: 1 to 4) defines spindle assignment x. Using the identification numbers of chuck, jaws and adapters (lathe center, etc.), you create the chucking equipment table. It is evaluated in the simulation (G65).

Enter chucking equipment data:

- ▶ „Select “Header – Chucking equipment”
- ▶ Position the cursor in the CHUCKING EQUIPMENT section.
- ▶ Press the INS key.
- ▶ Edit the “Chucking equipment” dialog box.

Edit the CHUCKING EQUIPMENT data:

- ▶ Position the cursor.
- ▶ Press ENTER
- ▶ Edit the “Chucking equipment” dialog box.

Parameters of the “Chucking equipment” dialog box:

- H: Number of chucking equipment (reference to G65).
- H=1: Chuck
 - H=2: Jaw
 - H=3: Adapter - spindle
 - H=4: Adapter - tailstock
- ID: Identification number of chucking equipment (reference to database).
- X: Gripping diameter of jaws
- Q: Chuck form of jaws (see G65)

4.4.4 Contour Definition

KONTUR [CONTOUR]

Assigns the following workpiece blank and finished part description to a contour.

Parameters

- Q: Number of the contour – 1..4
- X, Z: Zero point shift (reference: machine zero)
- V: Position of the coordinate system
- 0: The machine coordinate system applies
 - 2: Mirrored machine coordinate system (Z direction opposite to the machine coordinate system)



The chucking equipment table is used for the simulation graphics – it does not influence the execution of the program.

Example: CHUCKING EQUIPMENT table

SPANNMITTEL 1 [CHUCKING EQUIPMENT]

H1 ID “KH250”

H2 ID “KBA250-77”

...

ROHTEIL [BLANK]

Program section for defining the contour of the blank part.

FERTIGTEIL [FINISHED PART]

Program section for defining the contour of the finished part. To define the finished part, use additional section codes such as FRONT, SURFACE, etc.

FRONT, REAR SIDE

designates contours on the front and rear face

Parameters

Z: Position of the contour on the front/rear face – default: 0

SURFACE

identifies contours on the lateral surface.

Parameters

X: Reference diameter of lateral-surface contours.

HILFSKONTUR [AUXILIARY CONTOUR]

designates further turning contours (intermediate contours).



For several independent contour definitions for drilling/milling, use the program section codes (FRONT END, REAR END, etc.).

Examples for section codes in the finished part definitions

. . .

ROHTEIL [BLANK]

N1 G20 X100 Z220 K1

FERTIGTEIL [FINISHED PART]

N2 G0 X60 Z-80

N3 G1 Z-70

. . .

STIRN Z-25 [FRONT]

N31 G308 P-10

N32 G402 Q5 K110 A0 Wi72 V2 XK0YK0

N33 G300 B5 P10 W118 A0

N34 G309

STIRN Z0 [FRONT]

N35 G308 P-6

N36 G307 XK0YK0 Q6 A0 K34.641

N37 G309

. . .

4.4.5 BEARBEITUNG [MACHINING]

Program section for machining a workpiece. This code **must** be programmed.

ENDE [END]

Ends your NC program. This code **must** be programmed. It replaces M30.

4.4.6 UNTERPROGRAMM [SUBPROGRAM]

If you define a subprogram within your NC program (within the same file), it is designated with SUBPROGRAM, followed by the name of the subprogram (max. 8 characters).

RETURN

ends your NC subprogram.

4.5 Geometry Commands

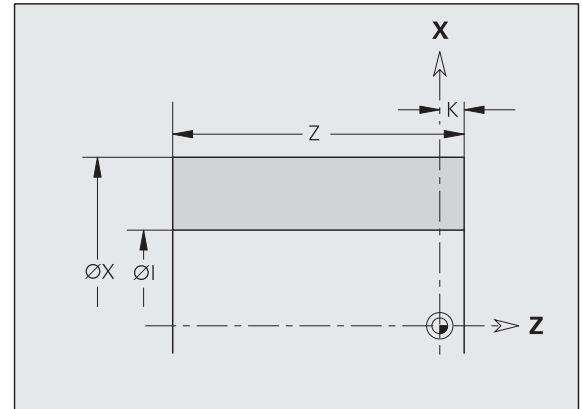
4.5.1 Definition of Blank

Chuck piece: bar/tube G20 Geo

Contour of a cylinder/hollow cylinder.

Parameters

- X: Diameter of a cylinder hollow cylinder
- Diameter of circumference of a polygonal blank
- Z: Length of blank
- K: Right edge (distance between workpiece zero point and right edge)
- I: Inside diameter for hollow cylinders



Cast part G21 Geo

Generates the workpiece blank contour from the finished part contour – plus the “equidistant allowance P”

Parameters

- P: Equidistant finishing allowance (reference: finished part contour)
- Q: Bore holes yes/no – default: Q=0
 - Q=0: Without bore holes
 - Q=1: With bore holes

4.5.2 Basic Contour Elements

Starting point of turning contour G0 Geo

Starting point of a turning contour.

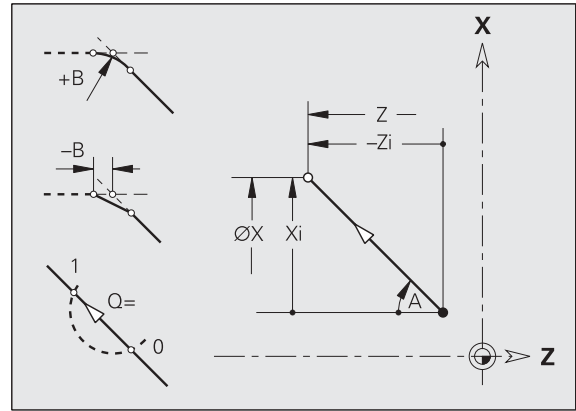
Parameters

- X, Z: Starting point of the contour (X diameter)

Line segment in a contour G1-Geo

Parameters

- X, Z: End point of contour element (X diameter)
- A: Angle to rotary axis – for angle direction see graphic support window
- Q: Selection of intersection – default: 0. End point, if the line segment intersects a circular arc.
 ■ Q=0: near intersection
 ■ Q=1: distance intersection
- B: Chamfer/rounding arc – transition to the next contour element. Program the theoretical end point when you enter a chamfer/rounding arc.
 ■ No entry in B: tangential transition
 ■ B=0: no tangential transition
 ■ B>0: Radius of the rounding arc
 ■ B<0: Width of chamfer
- E: Special feed factor for chamfer/rounding arc in a finishing cycle ($0 < E \leq 1$) – default: 1
 (special feed rate = active feed rate * E)



Programming X, Z: Absolute, incremental, modal or “?”

Circular arc in a contour

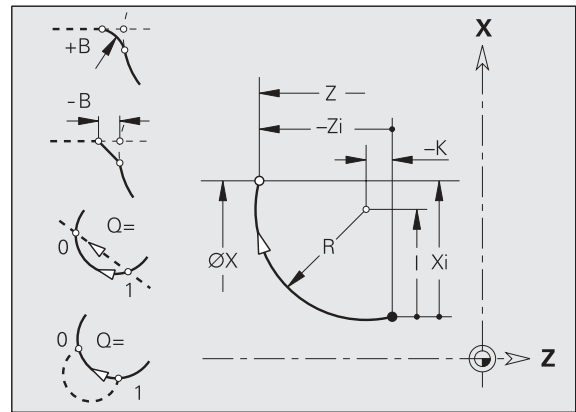
G2/G3 Geo – incremental center coordinates

G12/G13 Geo – absolute center coordinates

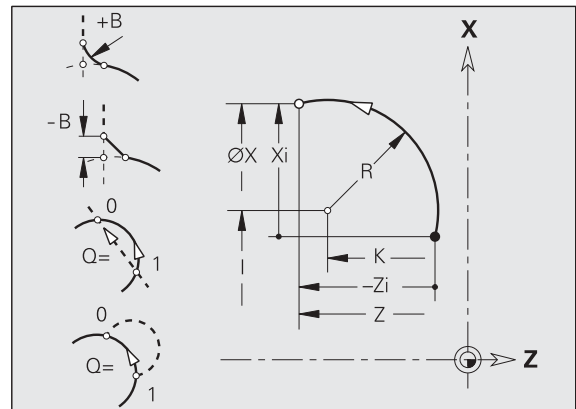
Direction of rotation: see help graphic

Parameters

- X, Z: End point of contour element (X diameter)
- R: Radius
- Q: Selection of intersection – default: 0. End point, if the circular arc intersects a circular arc.
 ■ Q=0: Far intersection
 ■ Q=1: Near intersection
- B: Chamfer/rounding arc – transition to the next contour element. Program the theoretical end point when you enter a chamfer/rounding arc.
 ■ No entry in B: tangential transition
 ■ B=0: no tangential transition
 ■ B>0: Radius of the rounding arc
 ■ B<0: Width of chamfer
- E: Special feed factor for chamfer/rounding arc in a finishing cycle ($0 < E \leq 1$) – default: 1
 (special feed rate = active feed rate * E)



G2 Geo



G13 Geo

G2/G3 – incremental center:

- I: Center (distance from starting point to center as radius)
- K: Center (distance from starting point to center)

G12/G13 – absolute center:

- I: Center (radius)
- K: Center



Programming X, Z: Absolute, incremental, modal or “?”

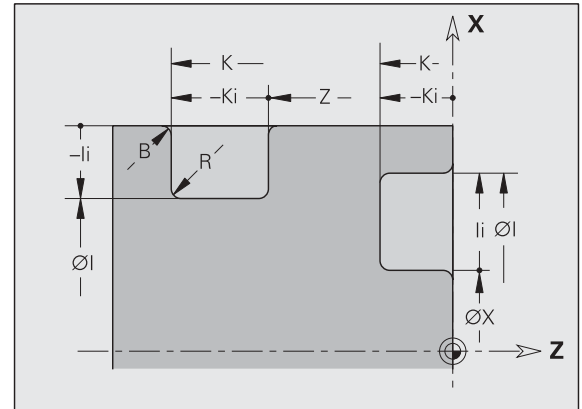
4.5.3 Contour Form Elements

Recess (Standard) G22 Geo

Recess on a paraxial reference element (G1). G22 is assigned to the previously programmed reference element.

Parameters

- X: Starting point of recess on the face (diameter)
Z: Starting point of recess on the lateral surface
I, K: Inside corner
■ I – recess on face: Recess end point (diameter value)
■ I – recess on lateral surface: Recess base (diameter value)
■ K – recess on face: Recess base
■ K – recess on lateral surface: Recess end point
Ii, Ki: Inside corner – incremental (pay attention to sign!)
■ Ii – recess on face: Recess width
■ Ii – recess on lateral surface: Recess depth
■ Ki – recess on face: Recess depth
■ Ki – recess on lateral surface: Recess end point (recess width)
B: Outside radius/chamfer (at both ends of the recess) – default: 0
■ B>0: Radius of rounding
■ B<0: Width of chamfer
R: Inside radius (in both corners of recess) – default: 0



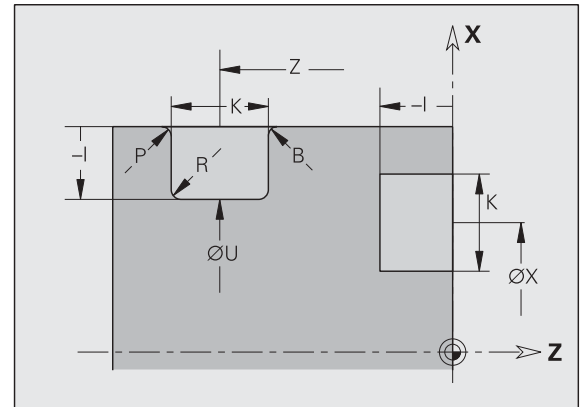
Program either "X" or "Z".

Recess (general) G23-Geo

Recess on a linear reference element (G1). G23 is assigned to the previously programmed reference element. On the lateral surface, the recess can be positioned on an inclined reference straight.

Parameters

- H: Recess type – default: 0
■ H=0: Symmetrical recess
■ H=1: relief turn
X: Center point of recess on the end face (diameter)
Z: Center point of recess on the lateral surface
I: Recess depth and position
■ I>0: Recess to right of reference element
■ I<0: Recess to left of reference element
K: Recess width (without chamfer/rounding)
U: Recess diameter (diameter of recess base) – use only if the reference element runs parallel to the Z axis
A: Recess angle – default: 0
■ With H=0: $0^\circ \leq A < 180^\circ$ (angle between edges of recess)
■ With H=1: $0^\circ < A \leq 90^\circ$ (angle between reference straight and recess edge)



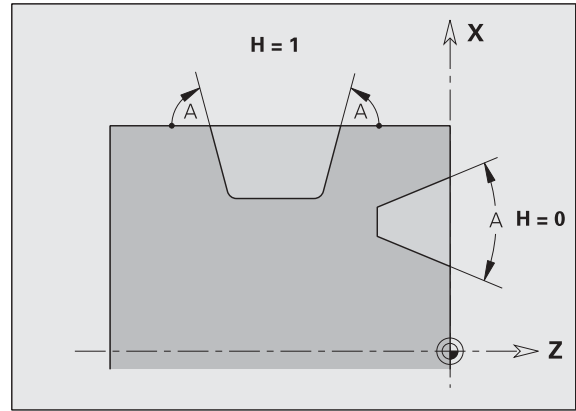
Simple recess

Continued ►

- B: Outside radius/chamfer; starting point near corner - default: 0
 ■ B>0: Radius of rounding
 ■ B<0: Width of chamfer
- P: Outside radius/chamfer; starting point distant from corner - default: 0
 ■ P>0: Radius of rounding
 ■ P<0: Width of chamfer
- R: Inside radius (in both corners of recess) – default: 0



The CNC PILOT refers the recess depth to the reference element. The recess base runs parallel to the reference element.



Recess or free rotation

Thread with undercut G24-Geo

Linear base element with linear thread (external or internal thread; metric ISO fine-pitch thread DIN 13 Part 2, Series 1) and a subsequent thread undercut (DIN 76).

Calling the contour macro:

N..G1 X..Z..B.. /Starting point for thread

N..G24 F..I..K..Z.. /Contours for thread and undercut

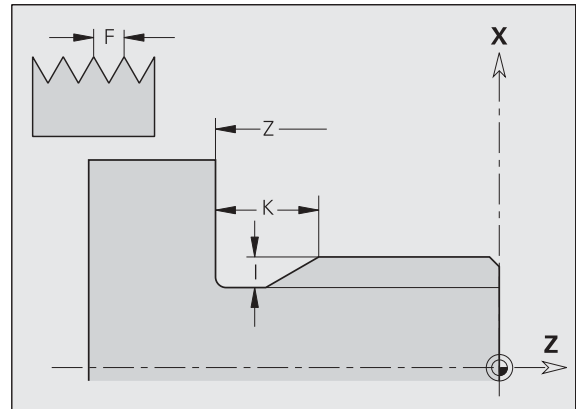
N..G1 X.. /Next surface element

Parameters

- F: Thread pitch
 I: Depth of undercut (radius)
 K: Width of undercut
 Z: End point of the undercut



- Use G24 only if the thread is cut in the definition direction of the contour.
- The thread is machined with G31.



Undercut contour G25-Geo

Generates the following undercut contours in paraxial inside contour corners. Program G25 after the first axis-parallel element.

Parameters

- H: Type of undercut – default: 0
- H=4: Undercut type U
 - H=0, 5: Undercut type DIN 509 E
 - H=6: Undercut type DIN 509 F
 - H=7: Thread undercut DIN 76
 - H=8: Undercut type H
 - H=9: Undercut type K

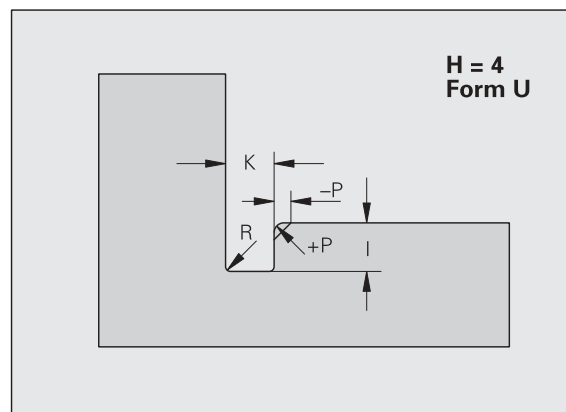
Calling the contour macro (example):

N..G1 Z..	/longitudinal element
N..G25 H..I..K..	/Undercut contour
N..G1 X..	/Next surface element

Undercut form U (H=4)

Parameters

- I: Depth of undercut (radius)
- K: Width of undercut
- R: Inside radius (in both corners of recess) – default: 0
- P: Outside radius/chamfer – default: 0
- P>0: Radius of rounding
 - P<0: Width of chamfer

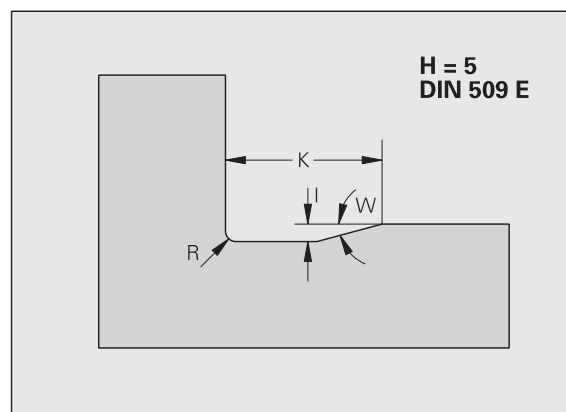


Undercut DIN 509 E (H=0, 5)

Parameters

- I: Depth of undercut (radius)
- K: Width of undercut
- R: Undercut radius (in both corners of the undercut)
- W: Undercut angle

If you do not enter parameters, the CNC PILOT calculates them from the diameter (see "11.1.2 Undercut Parameters DIN 509 E").



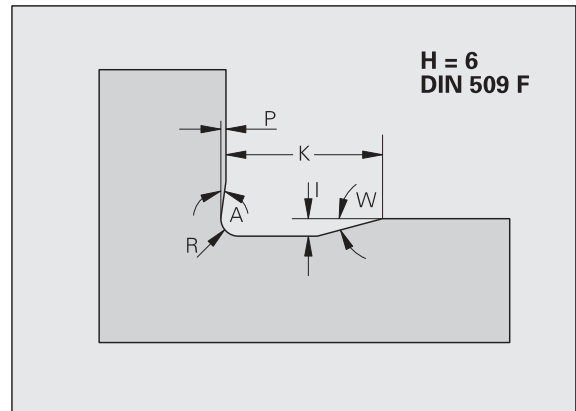
Continued ►

Undercut DIN 509 F (H=6)

Parameters

- I: Depth of undercut (radius)
- K: Width of undercut
- R: Undercut radius (in both corners of the undercut)
- P: transverse depth
- W: Undercut angle
- A: Transverse angle

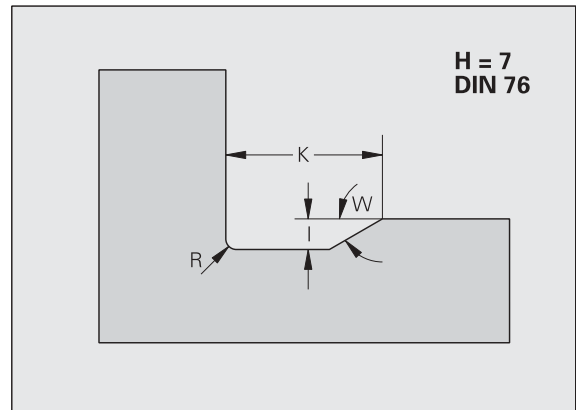
If you do not enter parameters, the CNC PILOT calculates them from the diameter (see "11.1.3 Undercut Parameters DIN 509 F").



Undercut DIN 76 (H=7)

Parameters

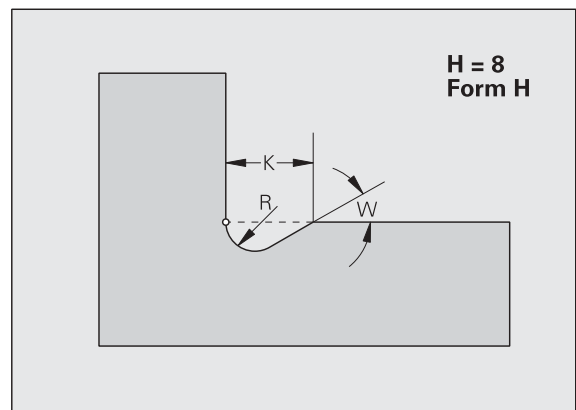
- I: Depth of undercut (radius)
- K: Width of undercut
- R: Undercut radius (in both corners of the undercut) – default: $R=0.6 \cdot I$
- W: Undercut angle – default: 30°



Undercut type H (H=8)

Parameters

- K: Width of undercut
- R: Undercut radius
- W: Plunging angle

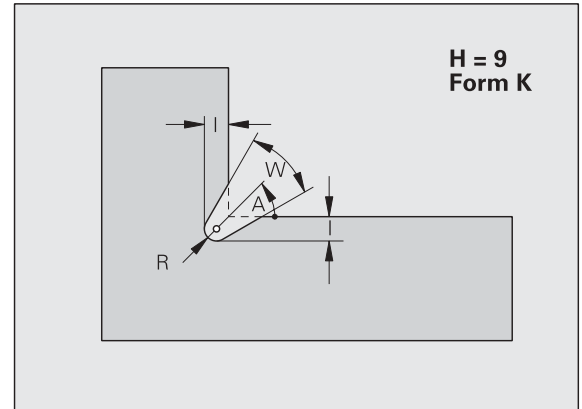


Continued ►

Undercut form K (H=9)

Parameters

- I: Undercut depth
R: Undercut radius – no value: The circular element is not machined
W: Undercut angle
A: Angle to linear axis – default: 45°



Thread (standard) G34-Geo

A simple or successive outside or inside thread (metric ISO fine-pitch thread DIN 13 Series 1). The CNC PILOT calculates all the required values.

Threads are interlinked by programming several G01/G34 blocks after each other.

Parameters

- F: Thread pitch – no value: Pitch from the standard table



- You need to program a linear contour element as a reference before G34 or in the NC block containing G34.
- The thread is machined with G31.

Thread (general) G37-Geo

Defines the different types of thread Multi-start threads and concatenated threads are possible. Threads are concatenated by programming several G01/G34 blocks after each other.

Parameters

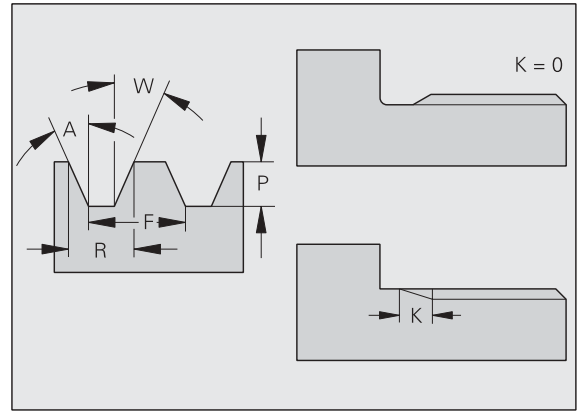
- Q: Type of thread – default: 1
- Q=1: Metric ISO fine-pitch thread (DIN 13 Part 2, Series 1)
 - Q=2: Metric ISO thread (DIN 13 Part 1, Series 1)
 - Q=3: Metric ISO taper thread (DIN 158)
 - Q=4: Metric ISO tapered fine-pitch thread (DIN 158)
 - Q=5: Metric ISO trapezoid thread (DIN 103 Part 2, Series 1)
 - Q=6: Flat metr. trapezoid thread (DIN 380 Part 2, Series 1)
 - Q=7: Metric buttress thread (DIN 513 Part 2, Series 1)
 - Q=8: Cylindrical round thread (DIN 405 Part 1, Series 1)
 - Q=9: Cylindrical Whitworth thread (DIN 11)
 - Q=10: Tapered Whitworth thread (DIN 2999)
 - Q=11: Whitworth pipe thread (DIN 259)
 - Q=12: Nonstandard thread
 - Q=13: UNC US coarse thread
 - Q=14: UNF US fine-pitch thread



- Program a linear contour element as a reference before G37.
- The thread is machined with G31.
- For standardized threads, the CNC PILOT defines the parameters P, R, A and W (see "11.1.4 Thread Parameters").
- Use Q=12 if you wish to use individual parameters.

Continued ►

- Q=15: UNEF US extra-fine-pitch thread
 - Q=16: NPT US taper pipe thread
 - Q=17: NPTF US taper dryseal pipe thread
 - Q=18: NPSC US cylindrical pipe thread with lubricant
 - Q=19: NPFS US cylindrical pipe thread without lubricant
- F: Thread pitch – required for Q=1, 3..7, 12. For other thread types, F is calculated from the diameter if it was not programmed (see "11.1.5 Thread Pitch").
- P: Thread depth – enter only for Q=12.
- K: Run-out length (for threads without undercut) – default: 0
- D: Reference point (position of thread run-out) – default: 0
- D=0: Run-out at end of reference element
 - D=1: Run-out at beginning of reference element
- H: Number of thread turns – default: 1
- A: Edge angle left – enter only for Q=12.
- W: Edge angle right – enter only for Q=12.
- R: Thread width – enter only for Q=12.
- E: Variable pitch (increases/reduces the pitch per revolution by E) – default: 0



Caution: Danger of collision!

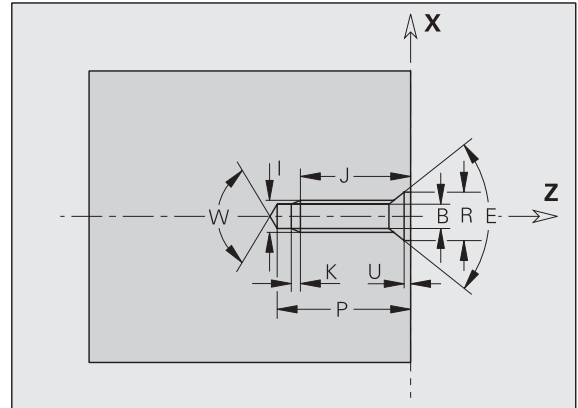
The thread is generated to the length of the reference element. Another linear element without undercut is to be programmed as overrun.

Bore hole (centric) G49 Geo

Single bore hole with countersink and thread **at the center of rotation** (front or rear face). The G49 hole is a form element, not part of the contour.

Parameters

- Z: Starting position for hole (reference point)
- B: Bore hole diameter
- P: Depth of hole (excluding point)
- W: Point angle – default: 180°
- R: Countersinking diameter
- U: Countersinking depth
- E: Countersinking angle
- I: Thread diameter
- J: Thread depth
- K: Thread runout length
- F: Thread pitch
- V: Left-hand or right-hand thread - default: 0
- V=0: Right-hand thread
 - V=1: Left-hand thread
- A: Angle (position of bore hole) – default: 0
- A=0: front end
 - A=180: tail end
- O: Centering diameter



- Program G49 in the FINISHED PART segment (not in FRONT or REAR SIDE).
- Machine the G49 hole with G71...G74.

4.5.4 Help Commands for Contour Definition

Overview	
G7	Precision stop ON
G8	Precision stop OFF
G9	Precision stop blockwise
G10	Influences finishing feed rate for total contour
G38	Influences finishing feed rate for basic contour elements block by block
G39	Only for form elements : <ul style="list-style-type: none">■ Influences finishing feed rate■ Additive compensation values■ Equidistant finishing allowances
G52	Equidistant oversize - blockwise
G95	Defines finishing feed rate for total contour
G149	Additive compensation values for basic contour elements

Precision stop ON G7-Geo

Switches "precision stop" on. It is a modal function. The block with G7 is run **with** precision stop. The CNC PILOT does not run the following block until the tool reaches the position tolerance window around the end point (for more on the tolerance window, see machine parameters 1106, 1156, ...).

Precision stop OFF G8-Geo

Switches the "precision stop" off. The block programmed with G8 is run **without** precision stop.

Blockwise precision stop G9 Geo

"Precision stop" for the NC block in which G9 is programmed (see also "G7 Geo").

Peak-to-valley height (surface texture) G10 Geo

Influences the finishing feed rate of G890.

Parameters

- H: Type of surface texture (see also DIN 4768)
- H=1: General roughness (profile depth) Rt1
 - H=2: Average roughness Ra
 - H=3: Mean roughness Rz
- RH: Peak-to-valley height (µm, inch mode: pinch)



- G10, G38, G52, G95 and G149 Geo apply for basic contour elements (G1, G2, G3, G12 and G13 Geo) – **not** for chamfers/rounding arcs that are programmed in connection with basic contour elements.
- The auxiliary commands of the contour description influence the finishing feed rate of the Cycles G869 and G890 – not the finishing feed rate in recessing cycles.



"Precision stop" is used for basic contour elements that are executed with G890 or G840.

Programming notes

- G10 Geo is modal.
- G95 Geo or G10 Geo without parameters switch the peak-to-valley height off.
- G10 RH... (without "H") overwrites the peak-to-valley height blockwise.
- G38 Geo overwrites the peak-to-valley height block by block.



The peak-to-valley height applies only to basic contour elements.

Feed rate reduction factor G38-Geo

Special feed rate for G890

Parameters

E: Special feed factor ($0 < E \leq 1$) – default: 1
(special feed rate = active feed rate * E)



The "special feed rate" is valid only for basic contour elements.

Programming notes

- G38 is a non-modal function.
- G38 is programmed **before** the contour element for which it is destined.
- G38 **replaces** another special feed rate or programmed peak-to-valley height.

Attributes for superimposed elements G39 Geo

Influences G890 in the **override elements (form elements)**:

- Chamfers/rounding arcs (for connecting basic elements)
- Undercuts
- Recesses

Machining factors influenced:

- Special feed rate
- Peak-to-valley height
- Additive D compensation
- Equidistant oversizes

Parameters

F: Feed per revolution

V: Type of surface texture (see also DIN 4768)

- V=1: General roughness (profile depth) Rt1
- V=2: Average roughness Ra
- V=3: Mean roughness Rz

RH: Peak-to-valley height (µm, inch mode: µinch)

D: Number of the additive compensation ($901 \leq D \leq 916$)

P: Finishing allowance (radius)

H: (Translation of P) absolute / additive – default: 0

- H=0: P replaces G57/G58 oversizes
- H=1: P is added to G57/G58 oversizes

E: Special feed factor ($0 < E \leq 1$) – default: 1
(special feed rate = active feed rate * E)

Programming notes

- G39 is a non-modal function.
- G39 is programmed **before** the contour element for which it is destined.
- G50 preceding a cycle (MACHINING section) cancels a finishing allowance programmed for that cycle with G39.



Only use peak-to-valley height ("V, RH"), finishing allowance ("F") and special feed rate ("E") alternately!

Blockwise oversize G52-Geo

Equidistant allowance that is taken into consideration in G810, G820, G830, G860 and G890.

Parameters

- P: Finishing allowance (radius)
H: (Translation of P) absolute / additive – default: 0
■ H=0: P replaces G57/G58 oversizes
■ H=1: P is added to G57/G58 oversizes

Programming notes

- G52 is a non-modal function.
- G52 is programmed **in** the NC block containing the contour element for which it is destined.
- G50 preceding a cycle (MACHINING section) cancels a finishing allowance programmed for that cycle with G52.

Feed per revolution G95-Geo

Influences the finishing feed rate of G890.

Parameters

- F: Feed per revolution

Programming notes

- G95 is a modal function.
- G10 resets a finishing feed rate set with G95.



- Use peak-to-valley height and finishing feed rate alternatively.
- The G95 finishing feed rate replaces a finishing feed rate defined in the machining program.

Additive compensation G149 Geo

The CNC PILOT manages 16 tool-independent compensation values.

To activate the additive compensation function, program G149 followed by a "D number" (for example, G149 D901). "G149 D900" resets the additive compensation function.

Parameters

- D: Additive compensation – default: D900 – range: 900 to 916

Programming notes

- Additive compensation is effective from the block in which G149 is programmed.
- An additive compensation remains effective until:
 - The next "G149 D900"
 - The end of the finished part description.



Note the direction of contour description!

4.5.5 Contour Position

Milling depth, contour position

You must define the "reference plane" or the "reference diameter" in the section code. Specify the depth and position of a milling contour (pocket, island) in the contour definition:

- With "depth P" in the previously programmed G308
- Alternatively on figures: Cycle parameter "depth P"

The **algebraic sign of "depth P"** defines the position of the milling contour (see table):

- P<0: Pocket
- P>0: Island

Section	P	Surface	Milling floor
FRONT	P<0	Z	Z+P
FRONT	P>0	Z+P	Z
REAR SIDE	P<0	Z	Z-P
REAR SIDE	P>0	Z-P	Z
SURFACE	P<0	X	X+(P*2)
SURFACE	P>0	X+(P*2)	X

X: Reference diameter from the section code
Z: Reference plane from the section code
P: "Depth" from G308 or from cycle parameter

Contours in More than One Plane

Programming with hierarchically nested contours:

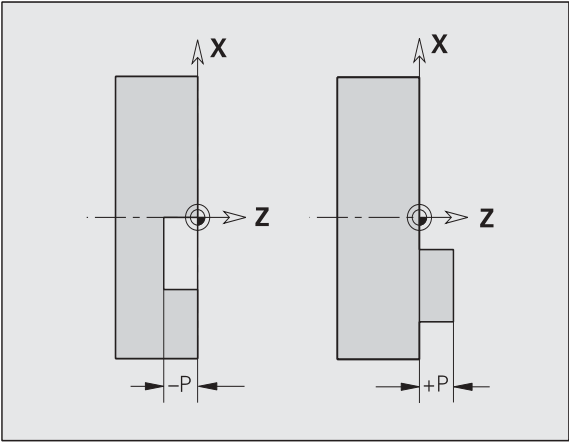
- Start with G308 "begin pocket/island" and end with G309 "End of pocket/island." G308 sets a "new" reference plane/reference diameter:
- The first G308 uses the reference plane defined in the section code.
- Every following G308 sets a new reference plane.
Calculation: Currently active reference plane + P (from the **previous** G308).
- G309 switches back to the previous reference plane.

Start pocket/island G308-Geo

New reference plane/reference diameter for hierarchically nested front-face, rear-face, or lateral face contours.

Parameters

P: Depth for pocket, height for islands



Pocket or island



Island: The area-milling cycles machine the complete area specified in the contour definition. Islands within this surface are not taken into consideration.

Continued ►

End pocket/island G309-Geo

End of a "reference plane." Every reference plane defined with G308 **must** be ended with G309!

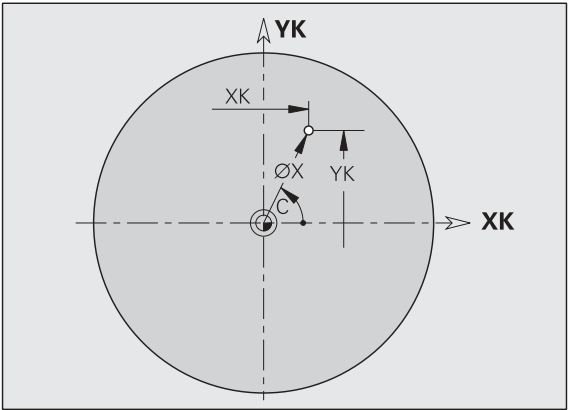
Example for "G308/G309"	
. . .	
FERTIGTEIL [FINISHED PART]	
. . .	
STIRN Z0 [FRONT]	Define reference plane
N7 G308 P-5	Beginning of "rectangle" with depth of -5
N8 G305 XK-5 YK-10 K50 B30 R3 A0	Rectangle
N9 G308 P-10	Beginning of "full circle in rectangle" depth -10
N10 G304 XK-3 YK-5 R8	Full circle
N11 G309	End of "full circle"
N12 G309	End of "rectangle"
MANTEL X100 [SURFACE]	Define reference diameter
N13 G311 Z-10 C45 A0 K18 B8 P-5	Linear slot with the depth "-5"
. . .	

4.5.6 Front and Rear Face Contours

Starting point of front/rear face contour G100 Geo

Parameters

- X: Starting point in polar coordinates (diameter)
- C: Starting point in polar coordinates (angular dimension)
- XK,YK: Starting point in Cartesian coordinates



Line segment in front-face/rear-face contour G101 Geo

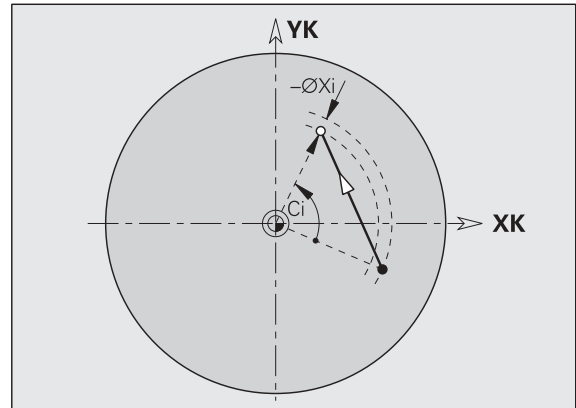
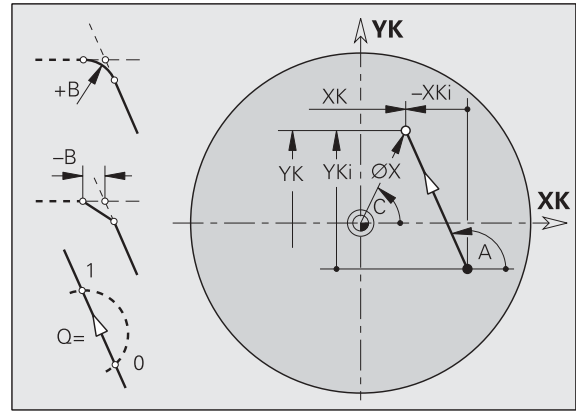
Parameters

- X: End point in polar coordinates (diameter)
 C: End point in polar coordinates (angular dimension)
 XK, YK: End point in Cartesian coordinates
 A: Angle to positive XK axis
 B: Chamfer/rounding arc – transition to the next contour element. Program the theoretical end point when entering a chamfer/rounding.
 ■ B No input: Tangential transition
 ■ B=0: Nontangential transition
 ■ B>0: Radius of rounding
 ■ B<0: Width of chamfer
 Q: Selection of intersection – default: 0. End point, if the line segment intersects a circular arc.
 ■ Q=0: near intersection
 ■ Q=1: distance intersection



Programming

- **X, XK, YK:** absolute, incremental, modal or "?"
 ■ **C:** absolute, incremental or modal

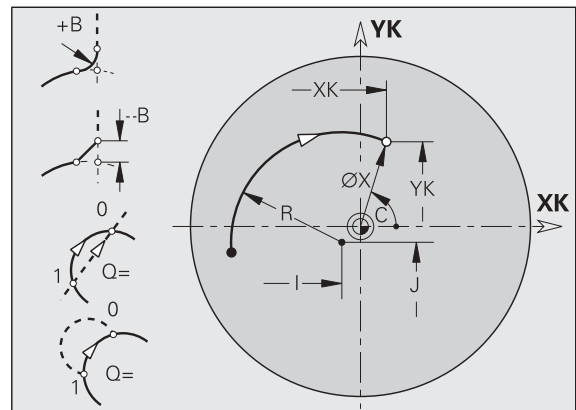


Circular arc on front/rear face G102/G103 Geo

Direction of rotation: see help graphic

Parameters

- X: End point in polar coordinates (diameter)
 C: End point in polar coordinates (angular dimension)
 XK, YK: End point in Cartesian coordinates
 R: Radius
 I, J: Center in Cartesian coordinates
 Q: Selection of intersection – default: 0. End point, if the circular arc intersects a circular arc.
 ■ Q=0: Far intersection
 ■ Q=1: Near intersection



G102 Geo

Continued ►

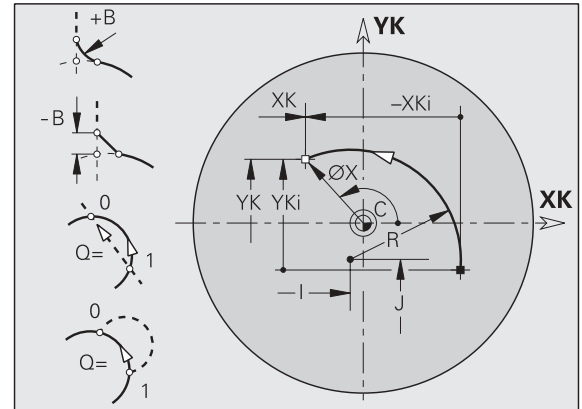
B: Chamfer/rounding arc – transition to the next contour element. Program the theoretical end point when you enter a chamfer/rounding arc.

- No entry in B: tangential transition
- B=0: no tangential transition
- B>0: Radius of the rounding arc
- B<0: Width of chamfer



Programming

- **X, XK, YK:** absolute, incremental, modal or "?"
- **C:** Absolute, incremental or modal
- **I, J:** Absolute or incremental
- End point must not be the starting point (no full circle).



G103 Geo

Hole on front/rear face G300 Geo

Hole with countersinking and thread.

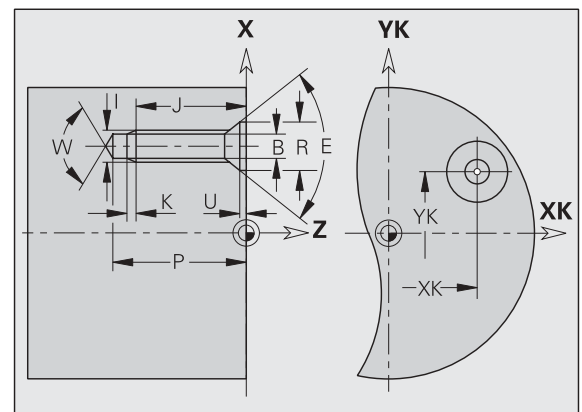
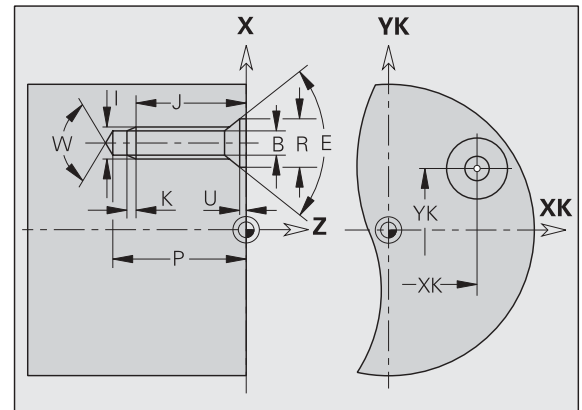
Parameters

XK, YK: Center in Cartesian coordinates

- B: Hole diameter
- P: Depth of hole (excluding point)
- W: Point angle – default: 180°
- R: Countersinking diameter
- U: Countersinking depth
- E: Countersinking angle
- I: Thread diameter
- J: Thread depth
- K: Thread runout length
- F: Thread pitch
- V: Left-hand or right-hand thread - default: 0
 - V=0: Right-hand thread
 - V=1: Left-hand thread
- A: Angle – inclination of hole (reference: Z axis)
 - Front face – default: 0° (range: -90° < A < 90°)
 - Rear face – default: 180° (range: 90° < A < 270°)
- O: Centering diameter



Machine the G300 holes with G71...G74.



Linear slot on face G301 Geo

Parameters

XK,YK: Center in Cartesian coordinates

A: Angle to longitudinal axis (reference: XK axis) – default: 0°

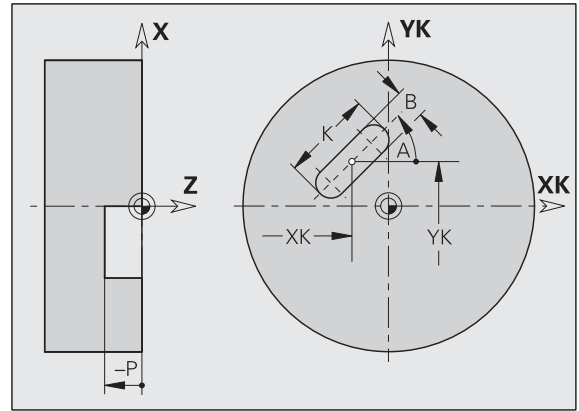
K: Slot length

B: Slot width

P: Depth/height – no entry: "P" from G308

■ P<0: Pocket

■ P>0: Island



Circular slot on front/rear face G302/G303 Geo

■ G302: Circular slot clockwise

■ G303: Circular slot counterclockwise

Parameters

I, J: Center of curvature in Cartesian coordinates

R: Curvature radius (reference: center point path of the slot)

A: Angle of starting point (reference: XK axis) – default: 0

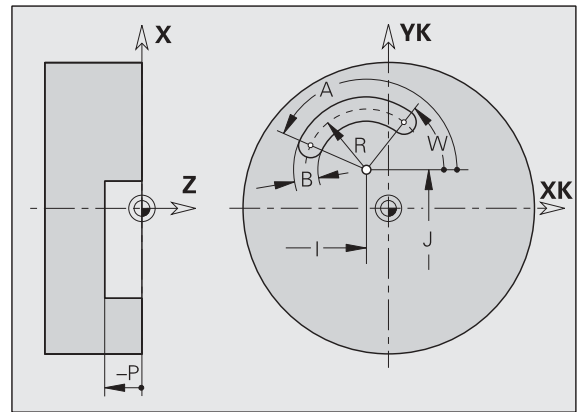
W: Angle of end point (reference: XK axis)

B: Slot width

P: Depth/height – no entry: "P" from G308

■ P<0: Pocket

■ P>0: Island



G302 Geo

Full circle on front/rear face G304 Geo

Parameters

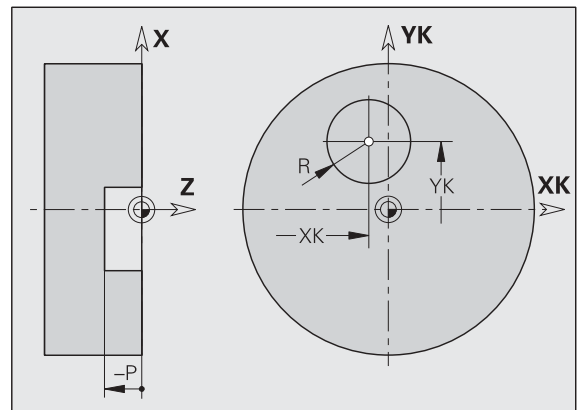
XK,YK: Circle center in Cartesian coordinates

R: Radius

P: Depth/height – no entry: "P" from G308

■ P<0: Pocket

■ P>0: Island



Rectangle on front/rear face G305 Geo

Parameters

XK, YK: Center in Cartesian coordinates

A: Angle to longitudinal axis (reference: XK axis) – default: 0°

K: Length

B: (height) width

R: Chamfer/rounding - default: 0

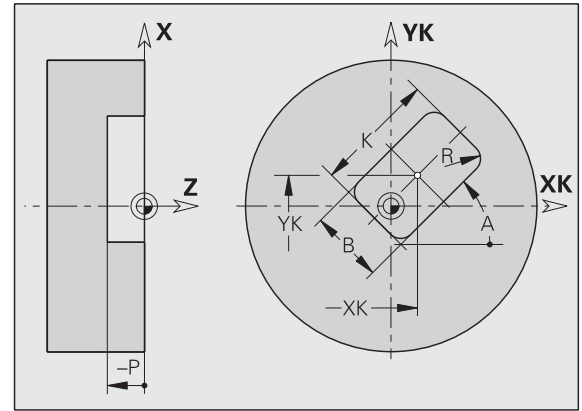
■ R>0: Radius of rounding

■ R<0: Width of chamfer

P: Depth/height – no entry: "P" from G308

■ P<0: Pocket

■ P>0: Island



Eccentric polygon on front/rear face G307 Geo

Parameters

XK, YK: Center in Cartesian coordinates

Q: Number of edges (Q>2)

A: Angle to a polygon side (reference: XK axis) – default: 0°

K: Edge length

■ K>0: Edge length

■ K<0: Key width (inside diameter)

R: Chamfer/rounding - default: 0

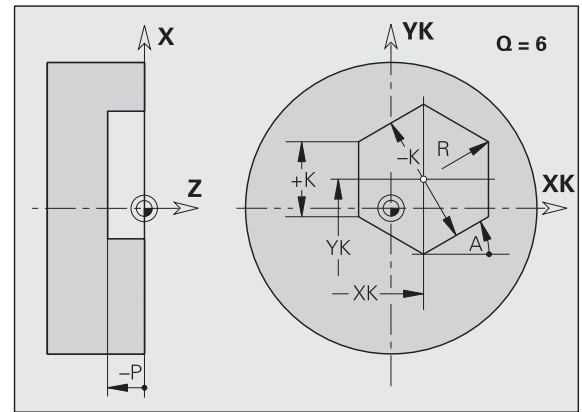
■ R>0: Radius of rounding

■ R<0: Width of chamfer

P: Depth/height – no entry: "P" from G308

■ P<0: Pocket

■ P>0: Island



Linear pattern on face G401 Geo

G401 is effective for the bore hole/figure defined in the following block (G300..305, G307).

Programming notes

■ Program the hole/figure in the following block without a center.

■ The milling cycle (MACHINING section) calls the hole/figure in the following block - not the pattern definition.

Parameters

Q: Number of holes/figures - default: 1

XK, YK: Starting point in Cartesian coordinates

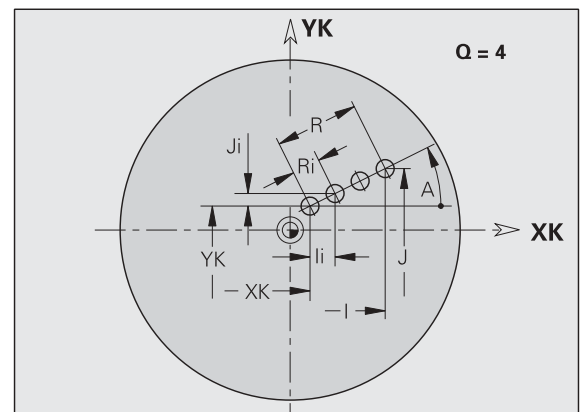
I, J: End point in Cartesian coordinates

li, Ji: Distance between figures (in XK, YK direction)

A: Angle to longitudinal axis (reference: XK axis) – default: 0°

R: Total length of pattern

Ri: Distance between figures (pattern distance)



Circular pattern on front/rear face G402 Geo

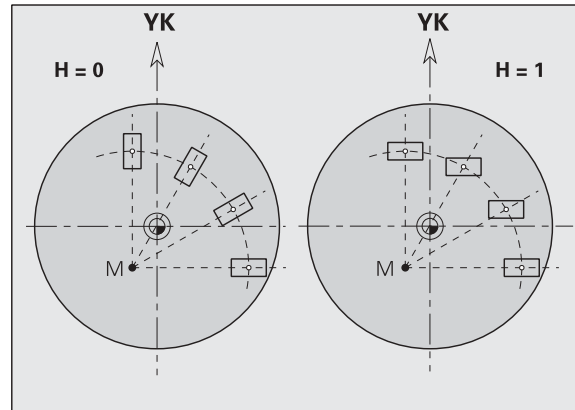
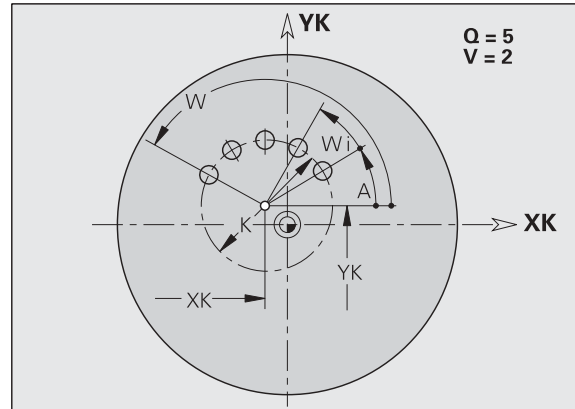
G402 is effective for the bore hole/figure defined in the following block (G300..305, G307).

Programming notes

- Program the hole/figure in the following block without a center. – Exception **circular slot**: the "center of curvature I, J" is added to the sample position (see " 4.5.8 Circular Pattern with Circular Slots").
- The milling cycle (MACHINING section) calls the hole/figure in the following block - not the pattern definition.

Parameters

- Q: Number of figures
- K: Pattern diameter
- A: Starting angle – position of the first figure (reference: XK axis) – default: 0°
- W: Ending angle – position of the last figure (reference: XK axis) – default: 360°
- Wi: Angle between figures
- V: Direction (orientation) – default: 0
- V=0 – without W: distribution over complete circle
 - V=0 – with W: distribution over long arc
 - V=0 – with Wi: algebraic sign of Wi defines the direction (Wi<0: clockwise)
 - V=1 – with W: clockwise
 - V=1 – with Wi: clockwise (sign of Wi has no meaning)
 - V=2 – with W: counterclockwise
 - V=2 – with Wi: counterclockwise (sign of Wi has no meaning)
- XK, YK: Center in Cartesian coordinates
- H: Position of figures – default: 0
- H=0: Normal position – figures are rotated about the circle center (rotation).
 - H=1: Original position – position of figure remains unchanged with respect to coordinate system (translation).



4.5.7 Lateral Surface Contours

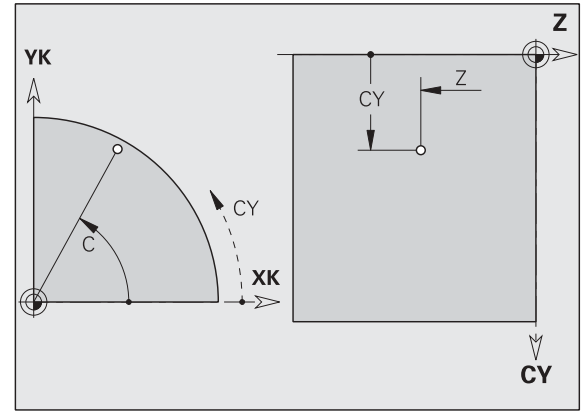
Starting point of lateral surface contour G110-Geo

Parameters

- Z: Starting point
C: Starting point (starting angle)
CY: Starting angle as linear value (referenced to unrolled reference diameter)



Program either Z, C or Z, CY.



Line segment in a lateral surface contour G111-Geo

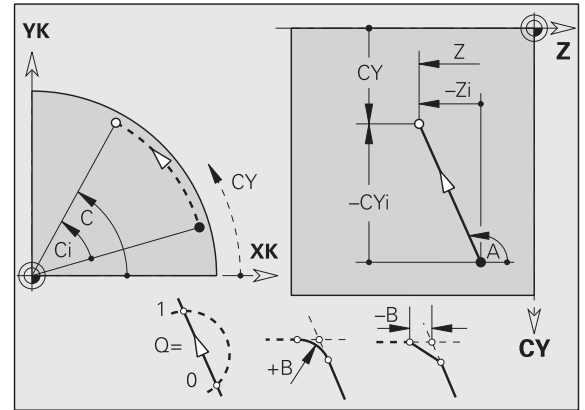
Parameters

- Z: End point
C: End point (end angle)
CY: End angle as linear value (referenced to unrolled reference diameter)
A: Angle (reference: positive Z axis)
B: Chamfer/rounding arc – transition to the next contour element. Program the theoretical end point when you enter a chamfer/rounding arc.
 - No entry in B: tangential transition
 - B=0: no tangential transition
 - B>0: Radius of the rounding arc
 - B<0: Width of chamfer
- Q: Selection of intersection – default: 0. End point, if the line segment intersects a circular arc.
 - Q=0: near intersection
 - Q=1: distance intersection



Programming

- Z, CY: Absolute, incremental, modal or “?”
- C: Absolute, incremental or modal
- Program either Z – C or Z – CY



Circular arc in lateral surface contour G112/G113-Geo

Direction of rotation: see help graphic

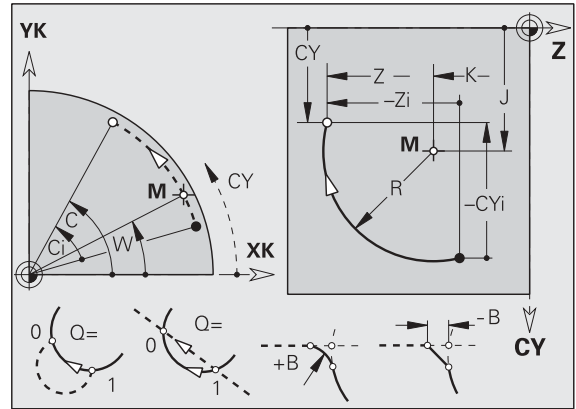
Parameters

- Z: End point
C: End point (end angle)
CY: End angle as linear value (referenced to unrolled reference diameter)
R: Radius
K: Center point (in Z direction)
W: Angle of the center point
J: Angle of the center point as a linear value
Q: Selection of intersection – default: 0. End point, if the circular arc intersects a circular arc.
 ■ Q=0: Far intersection
 ■ Q=1: Near intersection
B: Chamfer/rounding arc – transition to the next contour element. Program the theoretical end point when you enter a chamfer/rounding arc.
 ■ No entry in B: tangential transition
 ■ B=0: no tangential transition
 ■ B>0: Radius of the rounding arc
 ■ B<0: Width of chamfer

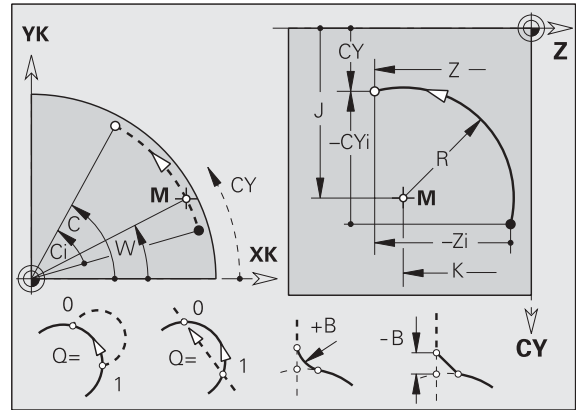


Programming

- **Z, CY:** Absolute, incremental, modal or "?"
- **C:** Absolute, incremental or modal
- **K, J:** Absolute or incremental
- Program either Z – C or Z – CY, and either K – W or K – J
- Program either "center" or "radius"
- With "radius": circular arcs possible only $\leq 180^\circ$



G112 Geo



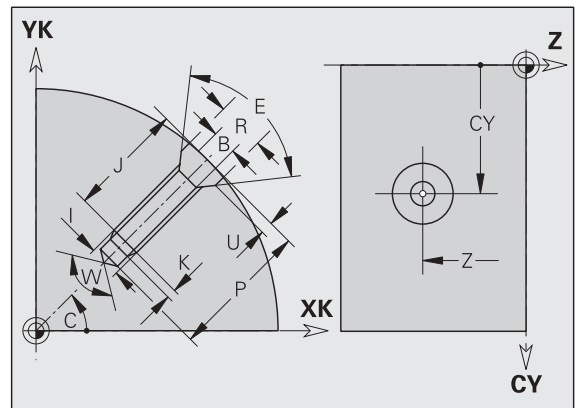
G113 Geo

Hole on lateral surface G310-Geo

Hole with countersinking and thread.

Parameters

- Z: Center (Z position)
C: Center (angle)
CY: Angle as linear value (referenced to unrolled reference diameter)
B: Hole diameter
P: Depth of hole (excluding point)
W: Point angle – default: 180°
R: Countersinking diameter
U: Countersinking depth



Continued ►

- E: Countersinking angle
- I: Thread diameter
- J: Thread depth
- K: Thread runout length
- F: Thread pitch
- V: Left-hand or right-hand thread - default: 0
 - V=0: Right-hand thread
 - V=1: Left-hand thread
- A: Angle (reference: Z axis) – default: 90° = vertical hole
(range: 0° < A < 180°)
- O: Centering diameter

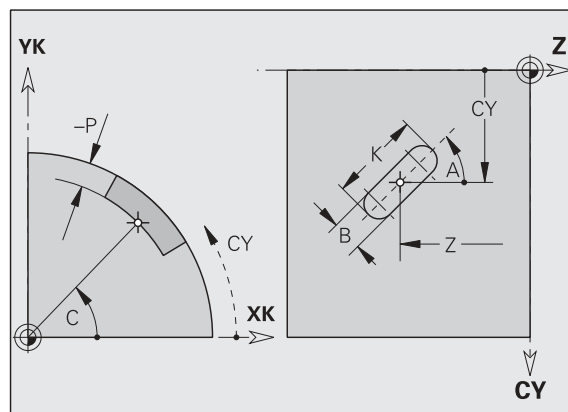


Machine the G310 hole with G71 ...G74.

Linear slot on lateral surface G311-Geo

Parameters

- Z: Center
- C: Center (angle)
- CY: Angle as linear value (referenced to unrolled reference diameter)
- A: Angle to longitudinal axis (reference: Z axis) – default: 0°
- K: Slot length
- B: Slot width
- P: Pocket depth – no entry: "P" from G308

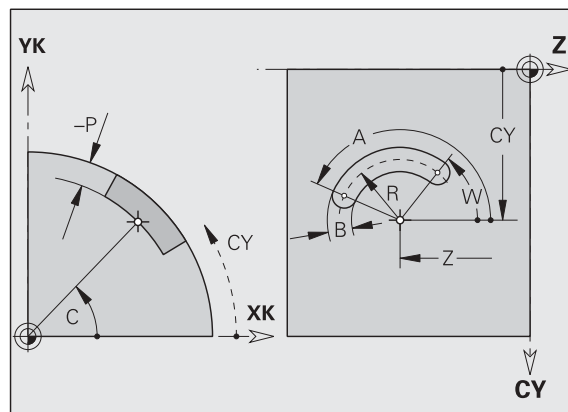


Circular slot on lateral surface G312/G313-Geo

- G312: Circular slot clockwise
- G313: Circular slot counterclockwise

Parameters

- Z: Center of curvature
- C: Center of curvature (angle)
- CY: Angle as linear value (referenced to unrolled reference diameter)
- R: Curvature radius (reference: center point path of the slot)
- A: Angle of starting point (reference: Z axis)
- W: Angle of end point (reference: Z axis)
- B: Slot width
- P: Pocket depth – no entry: "P" from G308

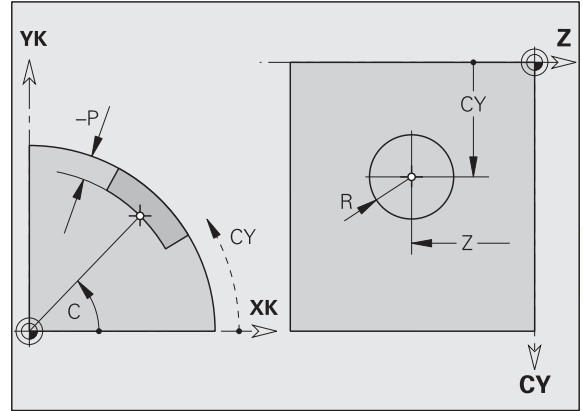


G312-Geo

Full circle on lateral surface G314-Geo

Parameters

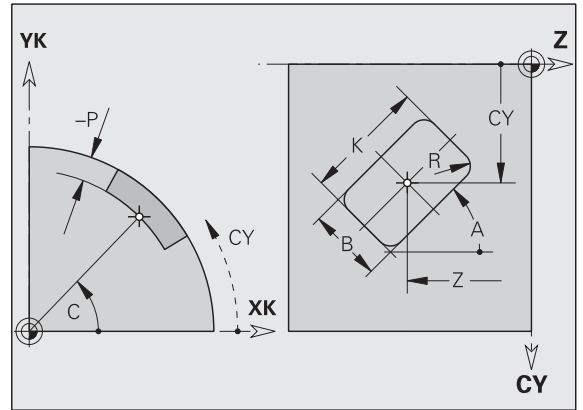
- Z: Circle center
 C: Circle center (angle)
 CY: Angle as linear value (referenced to unrolled reference diameter)
 R: Radius
 P: Pocket depth – no entry: "P" from G308



Rectangle on lateral surface G315-Geo

Parameters

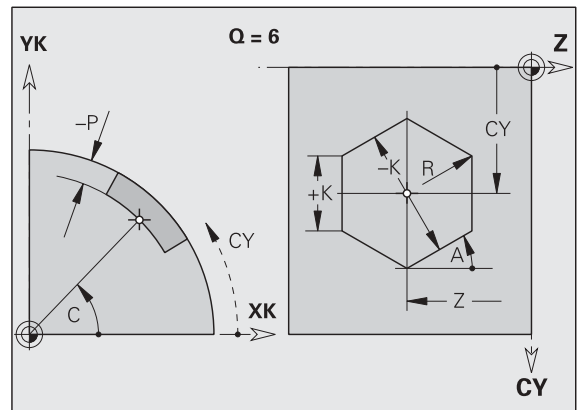
- Z: Center
 C: Center (angle)
 CY: Angle as linear value (referenced to unrolled reference diameter)
 A: Angle to longitudinal axis (reference: Z axis) – default: 0°
 K: Length
 B: Width
 R: Chamfer/rounding - default: 0
 ■ R>0: Radius of rounding
 ■ R<0: Width of chamfer
 P: Pocket depth – no entry: "P" from G308



Eccentric polygon on lateral surface G317-Geo

Parameters

- Z: Center
 C: Center (angle)
 CY: Angle as linear value (referenced to unrolled reference diameter)
 Q: Number of edges (Q>2)
 A: Angle to a polygon side (reference: Z axis) – default: 0°
 K: Edge length
 ■ K>0: Edge length
 ■ K<0: Key width (inside diameter)
 R: Chamfer/rounding - default: 0
 ■ R>0: Radius of rounding
 ■ R<0: Width of chamfer
 P: Pocket depth – no entry: "P" from G308



Linear pattern on lateral surface G411-Geo

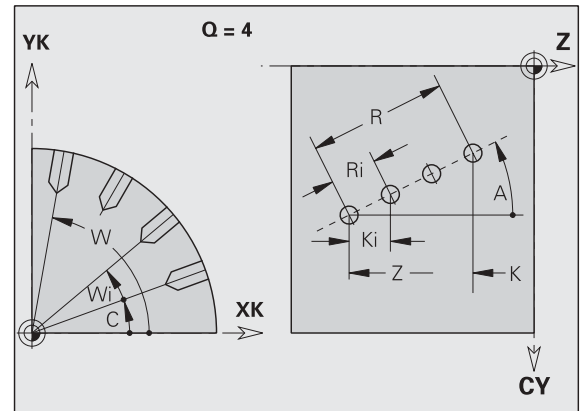
G411 is effective for the bore hole/figure defined in the following block (G310..315, 317).

Programming notes

- Program the hole/figure in the following block without a center.
- The milling cycle (MACHINING section) calls the hole/figure in the following block - not the pattern definition.

Parameters

- Q: Number of holes/figures - default: 1
 Z: Starting point
 C: Starting point (starting angle)
 K: End point
 W: End point (end angle)
 Ki: Distance between figures (in Z direction)
 Wi: Angular distance between figures
 A: Angle to longitudinal axis (reference: Z axis) – default: 0°
 R: Total length of pattern
 Ri: Distance between figures (pattern distance)



If you program Q, Z and C, the bore holes/figures will be ordered in a regular manner along the circumference.

Circular pattern on lateral surface G412-Geo

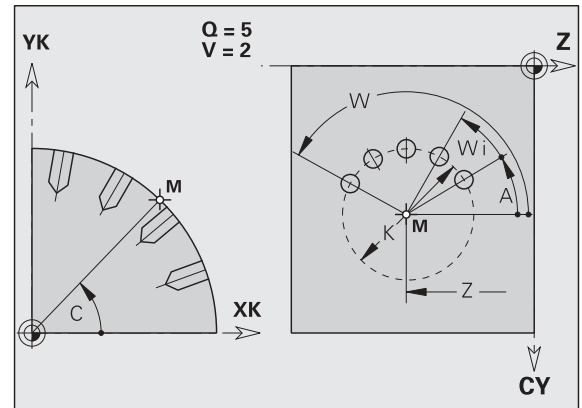
G412 is effective for the bore hole/figure defined in the following block (G310..315, 317).

Programming notes

- Program the hole/figure in the following block without a center. – Exception **circular slot**: the "center of curvature I, J" is added to the sample position (see " 4.5.8 Circular Pattern with Circular Slots").
- The milling cycle (MACHINING section) calls the hole/figure in the following block - not the pattern definition.

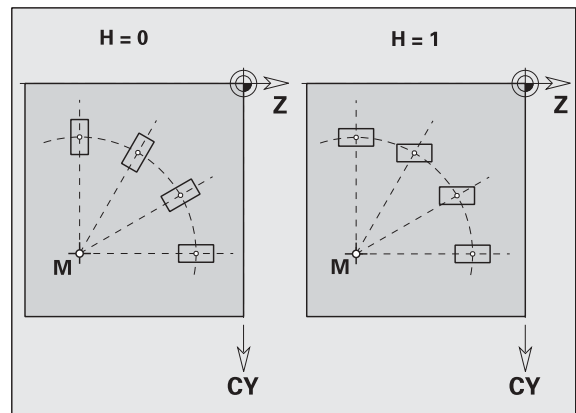
Parameters

- Q: Number of figures
 K: Diameter of circle
 A: Starting angle – position of the first figure (reference: Z axis) – default: 0°
 W: End angle – position of the last figure (reference: Z axis) – default: 360°
 Wi: Distance between figures



Continued ►

- V: Direction (orientation) – default: 0
- V=0 – without W: distribution over complete circle
 - V=0 – with W: distribution over long arc
 - V=0 – with Wi: algebraic sign of Wi defines the direction (Wi<0: clockwise)
 - V=1 – with W: clockwise
 - V=1 – with Wi: clockwise (sign of Wi has no meaning)
 - V=2 – with W: counterclockwise
 - V=2 – with Wi: counterclockwise (sign of Wi has no meaning)
- Z: Center of pattern
- C: Center of pattern (angle)
- H: Position of figures – default: 0
- H=0: Normal position – figure is rotated about the circle center (rotation).
 - H=1: Original position – position of figure remains unchanged with respect to the coordinate system (translation).



4.5.8 Circular Pattern with Circular Slots

In addition to the pattern positions, for circular patterns you program the center and radius of curvature. DIN PLUS and TURN PLUS calculate the position of the slots from the center of the pattern and center of curvature:

- If pattern center = center of curvature **and** pattern radius = radius of curvature, **then:** Pattern position = center of slot center line
- Pattern center≠center of curvature **or** pattern radius≠ center of curvature, **then:** Pattern position = center of curvature

The following rules apply for the orientation of the slots (pattern definition):

- **Normal position:** The starting/end angles are defined **relative** to the pattern positions. (The orientation angle is added to the starting/end angle.)
- **Original position:** Starting/end angles are defined **absolutely**.

The following examples and figures show the programming of a circular pattern with circular slots.

Example for a slot center as reference and normal position:

. . .	Slots arranged at distance of "pattern radius"
N7 G472 Q4 K30 A0 X0 Y0 H0	around the pattern center point.
N8 G373 X0 Y0 R15 A-20 W20 B3 P1	

Example for a slot center as reference and original position:

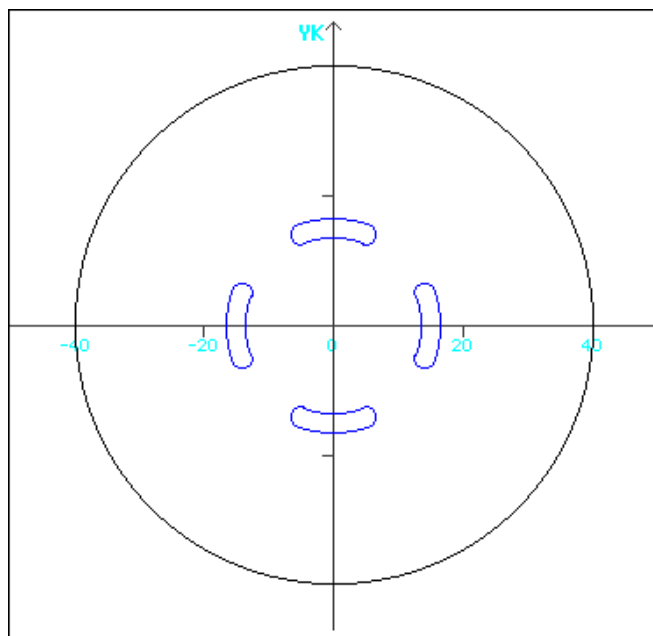
. . .	All slots are located at the same position
N7 G472 Q4 K30 A0 X0 Y0 H1	(center of curvature = center of pattern)
N8 G373 X0 Y0 R15 A-20 W20 B3 P1	

Example of a center of curvature as reference and normal position:

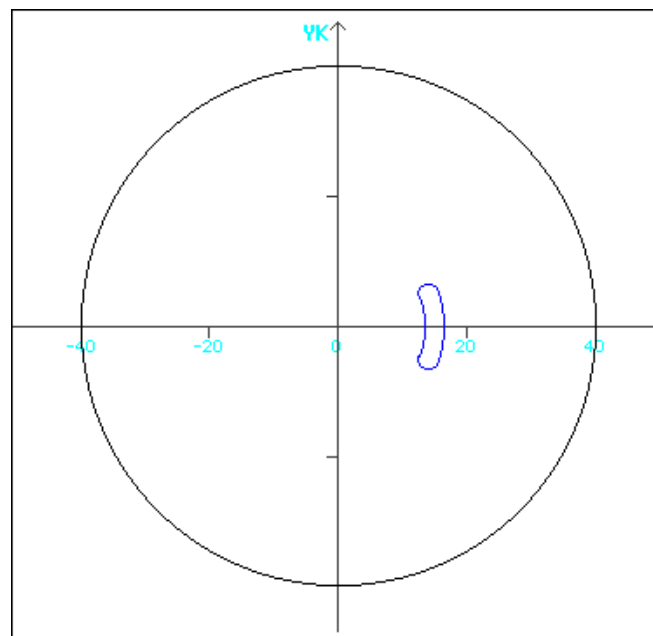
. . .	Slots arranged at distance of "pattern radius+
N7 G472 Q4 K30 A0 X5 Y5 H0	curvature radius" around the pattern center
N8 G373 X0 Y0 R15 A-20 W20 B3 P1	(pattern center point: X=5;Y=5).

Example for a center of curvature as reference and original position:

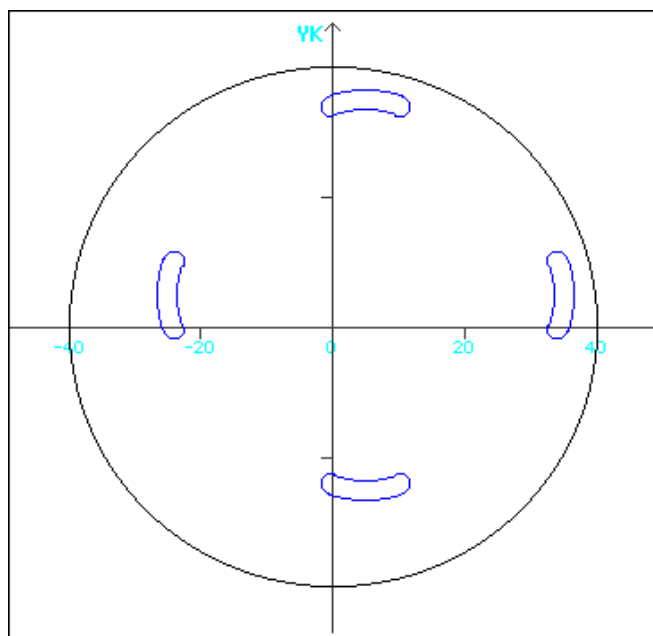
. . .	Slots are arranged at the distance of the "pattern radius"
N7 G472 Q4 K30 A0 X5 Y5 H1	curvature radius" around the pattern center
N8 G373 X0 Y0 R15 A-20 W20 B3 P1	while retaining the starting/end angles
	(pattern center point: X=5;Y=5).



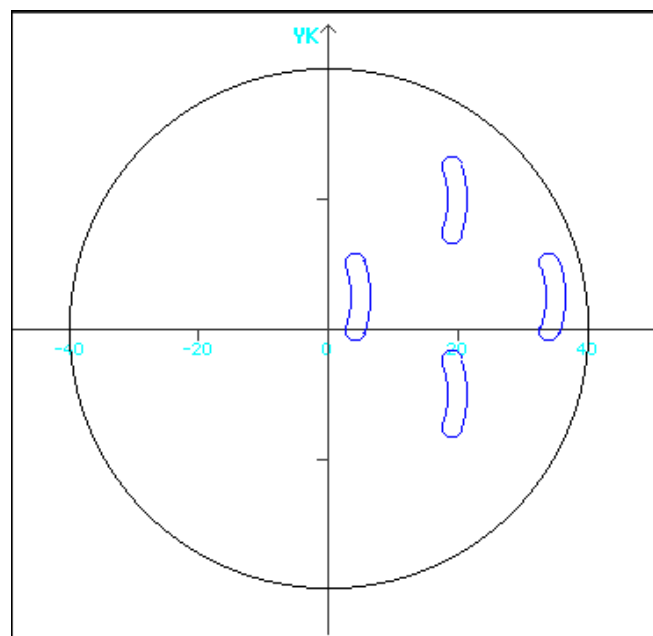
Example for a slot center line as reference and normal position



Example for a slot center as reference and original position



Example for a center of curvature as reference and normal position



Example for a center of curvature as reference and original position

4.6 Machining Commands

4.6.1 Assigning the Contour to the Operation

Workpiece group G99

If more than one contour description is defined in an NC program (workpieces), use G99 to assign the "contour Q" to the following machining sequence. The slide code before the NC block defines the slides that machine this contour. If G99 was not yet programmed (for example at the start of the program), all slide work in "contour 1."

Parameters

Q: Tool number – specified in CONTOUR
D: Spindle number – spindle that holds the workpiece
X, Z: Zero point shift (reference: machine zero)



- The simulation
 - positions the tool using the "shift in X, Z"
 - calculates and positions the chuck using the "spindle number D" (G99 does not replace the G65)
- Program G99 again if the workpiece is transferred to another spindle and/or moves its position in the working space.

4.6.2 Tool Positioning without Machining

Rapid traverse G0

The tool moves at rapid traverse along the shortest path to the target point

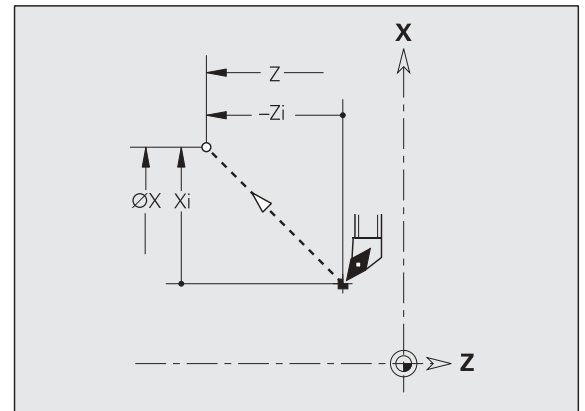
Parameters

X, Z: Diameter, length to target point (X diameter)



Programming X, Z: Absolute, incremental or modal

Programming in the Y axis: See "CNC PILOT 4290 with Y Axis" User's Manual."



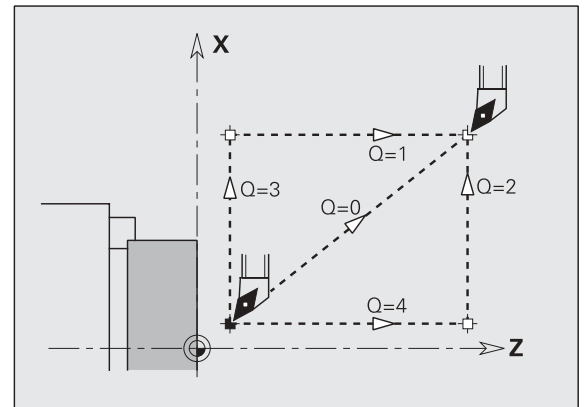
Approach to tool change position G14

The slide moves at rapid traverse to the tool change position. In setup mode, define permanent coordinates for the tool change.

Parameters

Q: Sequence – default: 0
0: diagonal path of traverse
1: First X direction, then Z
2: First Z direction, then X
3: Only X direction
4: Only Z direction

Programming in the Y axis See "CNC PILOT 4290 with Y Axis" User's Manual.



Rapid traverse to machine coordinates G701

The slide moves at rapid traverse on the shortest path to the target point.

Parameters

X, Z: End point (X diameter value)

Programming in the Y axis See "CNC PILOT 4290 with Y Axis" User's Manual.



"X, Z" refer to the machine zero point and the slide zero point.

4.6.3 Simple Linear and Circular Movements

Linear path G1

The tool moves linearly at the feed rate to the "end point."

Parameters

X, Z: Diameter, length to end point (X diameter)

A: Angle (angular direction: see graphic support window)

Q: Selection of intersection – default: 0. End point, if the line segment intersects a circular arc.

■ Q=0: near intersection

■ Q=1: distance intersection

B: Chamfer/rounding arc – transition to the next contour element. Program the theoretical end point when you enter a chamfer/rounding arc.

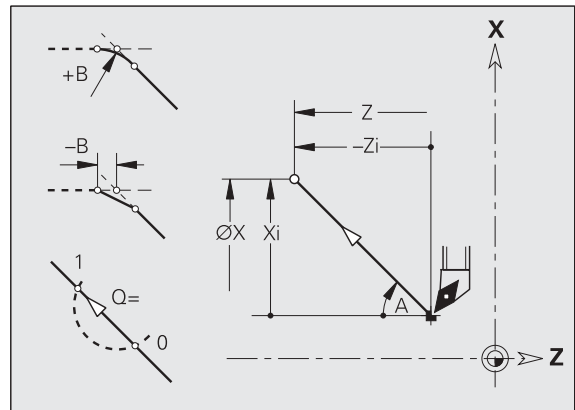
■ No entry in B: tangential transition

■ B=0: no tangential transition

■ B>0: Radius of the rounding arc

■ B<0: Width of chamfer

E: Special feed factor for chamfer/rounding ($0 < E \leq 1$) – default: 1
(special feed rate = active feed rate * E)



Programming X, Z: Absolute, incremental, modal or "?"

Programming in the Y axis See "CNC PILOT 4290 with Y Axis" User's Manual.

Circular paths

G2, G3 – incremental center coordinates

G12, G13 – absolute center coordinates

The tool moves in a circular arc at the feed rate to the "end point."

Direction of rotation: see help graphic.

Parameters

X, Z: Diameter, length to end point (X diameter)

R: Radius ($0 < R \leq 200\,000$ mm)

Q: Selection of intersection – default: Q=0. End point, if the circular arc intersects a circular arc.

- $Q=0$: Far intersection
- $Q=1$: Near intersection

B: Chamfer/rounding arc – transition to the next contour element.
Program the theoretical end point when you enter a chamfer/
rounding arc.

- No entry in B: tangential transition
- $B=0$: no tangential transition
- $B>0$: Radius of the rounding arc
- $B<0$: Width of chamfer

E: Special feed factor for chamfer/rounding
($0 < E \leq 1$) – default: 1
(special feed rate = active feed rate * E)

G2, G3 – incremental center :

I, K: Center (distance from starting point to center; I radius)

G12, G13 – center absolute:

I, K: Center (I radius)

Programming in the Y axis See "CNC PILOT 4290 with Y Axis" User's Manual.

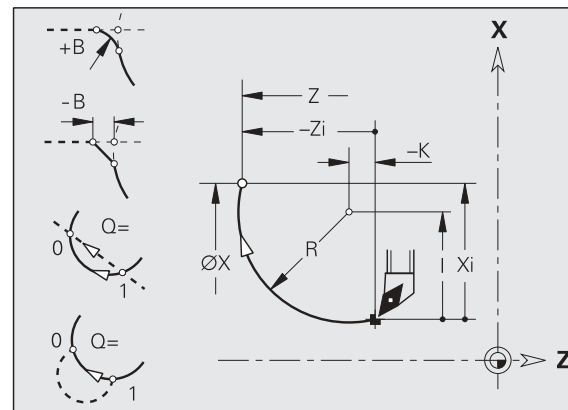


Programming X, Z: Absolute, incremental, modal or "?"

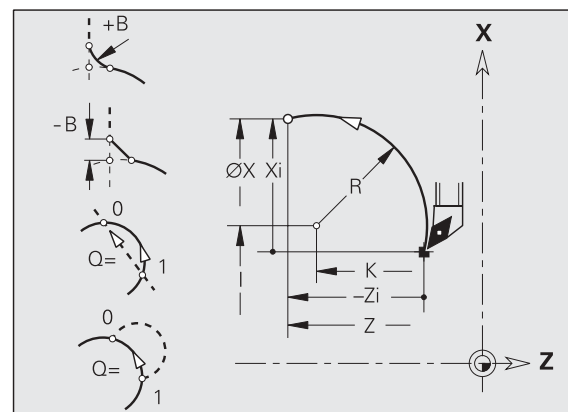


Danger of collision!

If V variables are used for calculating the address parameters, a limited contour check is carried out. Ensure that the variable values produce a circular arc.



Circular arc G2



Circular arc G13

4.6.4 Feed Rate and Spindle Speed

Speed limitation Gx26

G26: Spindle; Gx26: Spindle x (x: 1...3)

The speed limit remains in effect until the end of the program or until a new value is programmed for G26/Gx26.

Parameters

S: (Maximum) speed



Actual S > "absolute maximum speed" (machine parameter 805, ff), applies to the parameter value.

Acceleration (slope) G48

Define the approach acceleration, braking acceleration, and maximum feed rate. G48 is a modal function.

If G48 is not programmed,

- Acceleration and deceleration values will be taken from machine parameter 1105, ... "Acceleration/deceleration of linear axis"
- Maximum feed rate: machine parameter 1101, ... "maximum axis velocity"

Parameters

- E: Acceleration starting an axis – default: Parameter value
 F: Acceleration for braking an axis – default: parameter value
 H: Programmed acceleration On/Off
 ■ H=0: Switch off programmed acceleration after next traverse
 ■ H=1: Switch on programmed acceleration
 P: Maximum feed rate – default: parameter value



- If P > parameter value, the parameter value applies.
- "E, F and P" refer to the X or Z axis. The acceleration/feed rate for the slide is not higher with axis-parallel traverses.

Interrupted feed G64

Briefly interrupts the programmed feed rate. G64 is a modal function.

- Switch-on: Program G64 with "E and F"
- Switch-off: Program G64 without parameter

Parameters

- E: Duration of pause (Interval time: range: 0.01 s < E < 99.99 s)
 F: Duration of feed rate (feed period: range: 0.01 s < E < 99.99 s)

Feed per minute for rotary axes G192

Feed rate when only a rotary axis (auxiliary axis) is moved.

Parameters

- F: Feed per minute (in °/min)

Feed per tooth Gx93

Drive-dependent feed rate with respect to the number of teeth on the milling cutter (x: spindle 1...3).

Parameters

F: Feed per tooth (mm/tooth or inch/tooth)



The actual value display shows the feed rate in mm/rev.

Constant feed G94 (feed per minute)

Drive-independent feed rate.

Parameters

F: Feed per minute (mm/min / inch/min)

Feed per revolution Gx95

G95: Spindle; Gx95: Spindle x (x: 1...3)

Drive-dependent feed rate.

Parameters

F: Feed per revolution (mm/rev / inch/rev)

Constant cutting speed Gx96

G96: Spindle; Gx96: Spindle x (x: 1...3)

The spindle speed is dependent on the X position of the tool tip or on the diameter of the driven tools.

Parameters

S: Cutting velocity (in m/min or ft/min)



G26/Gx26 limits the spindle speed.

Speed Gx97

G97: Spindle; Gx97: Spindle x (x: 1...3)

Constant spindle speed.

Parameters

S: Speed (in revolutions per minute)

4.6.5 Cutter Radius Compensation (TRC/MCRC)

Tooth and cutter radius compensation (TRC)

If TRC is not used, the theoretical tool tip is the reference point for the paths of traverse. This might lead to inaccuracies when the tool moves along non-paraxial paths of traverse. The TRC function corrects programmed paths of traverse (see section "1.5 Tool Dimensions").

With Q=0, the TRC **reduces** the feed rate at arcs (G2, G3, G12, G13) and rounding arcs if the "shifted radius < original radius". The "special feed rate" is corrected when a rounding as transition to the next contour element is machined.

Reduced feed rate:

Feed rate * (offset radius / original radius)

Milling cutter radius compensation (MCRC)

Without the MCRC function, the system defines the center of the cutter as the zero point for the paths of traverse. With the TRC function, the CNC PILOT accounts for the outside cutting radius when moving along the programmed paths of traverse (see "1.5 Tool Dimensions").

Recessing, area clearance and milling cycles already include TRC/MCRC calls. You must therefore ensure that TRC/MCRC is disabled before you call these cycles. There are a few exceptions to this rule that will be described where concerned.

G40: Switch off TRC/MCRC

- The TRC is effective up to the block before G40
- In the block with G40 or in the block after G40, only a line is permissible (G14 is not allowed)

G41/G42: Switch on TRC/MCRC

- A straight line segment (G0/G1) must be programmed in the block containing G41/G42 or after the block containing G41/G42
- The TRC/MCRC is effective beginning with the next positioning command

G41: Switch on TRC/MCRC – compensation of the tool-tip/cutter radius **to the left of the contour in traverse direction.**

G42: Switch on TRC/MCRC – compensation of the tool-tip/cutter radius **to the right of the contour in traverse direction.**

Parameters (G41/G42)

- Q: Machining plane – default: 0
- Q=0: TRC on the turning plane (X-Z plane)
 - Q=1: MCRC on the face (X-C plane)
 - Q=2: TRC on the lateral surface (Z-C plane)
 - Q=3: TRC on the ace (X-Y plane)
 - Q=4: TRC on the lateral surface (Y-Z plane)
- H: Output (only with MCRC) – default: 0
- H=0: Intersecting areas which are programmed in directly successive contour elements are not machined.
 - H=1: The complete contour is machined – even if certain areas are intersecting.
- O: Feed rate reduction – default: 0
- O=0: Feed rate reduction active
 - O=1: No feed rate reduction



- If the tool radii are larger than the contour radii, the TRC/MCRC might cause endless loops. **Recommendation:** Use the finishing cycle G890 / milling cycle G840.
- Never select MCRC during a perpendicular approach to the respective plane.
- Remember when calling **subprograms** with "active TRC/MCRC":
Switch the TRC/MCRC off
– in the subprogram in which it was switched on
– in the main program if it was switch on there.

	Function of the TRC/MCRC
. . .	
N.. G0 X10 Z10	
N.. G41 G0 Z20	Path of traverse: from X10/Z10 to X10+TRC/Z20+TRC
N.. G1 X20	The path of traverse is "shifted" by the TRC
N.. G40 G0 X30 Z30	Path of traverse from X20+TRC/Z20+TRC to X30/Z30
. . .	

4.6.6 Zero Point Shift

You can program several zero shifts in one NC program. The relationship of the coordinates for blank/finished part, auxiliary contours are retained by the zero offset description.

G920 temporarily deactivates zero point shifts – G980 reactivates them.

Zero point shift G51

Shifts the workpiece zero point by "Z" (or "X"). The shift is referenced to the workpiece zero point defined in setup mode.

Even if you shift the zero point several times with G51, it is still always referenced to the workpiece zero point defined in setup mode.

The zero point shift remains in effect up to the end of the program or until it is canceled by another zero point shift.

Parameters

X, Z: Displacement (X radius value) – default: 0

Parameter-dependent zero offset G53, G54, G55

Shifts the workpiece zero point by the value defined in the setup parameters 3, 4, 5. The shift is referenced to the workpiece zero point defined in setup mode.

Even if you shift the zero point several times with G53, G54, G55, it is still always referenced to the workpiece zero point defined in setup mode.

The zero shift applies until the end of the program or until it is canceled by another zero shift



A shift in X is entered as a radius.

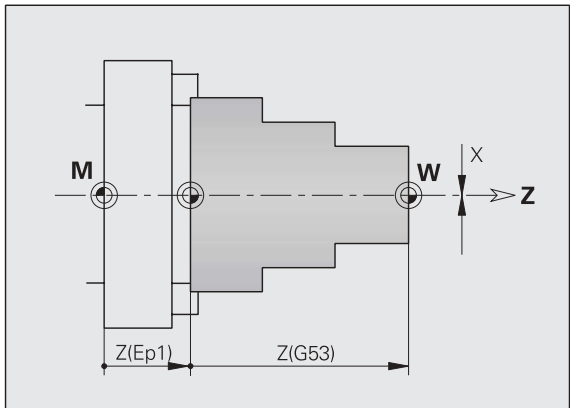
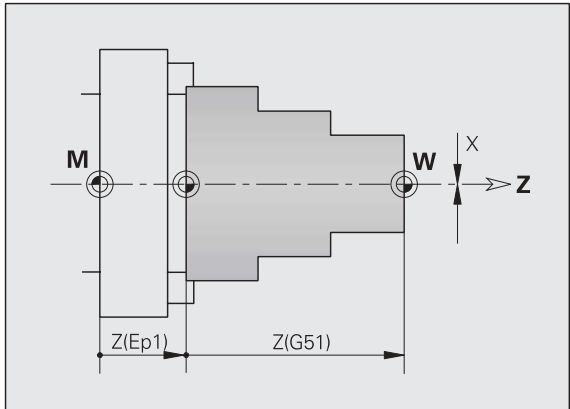
Overview

G51 ■ Relative shift
 ■ Programmed shift
 ■ Reference: Previously defined workpiece zero point

G53, G54, G55 ■ Relative shift
 ■ Shift defined in parameters
 ■ Reference: Previously defined workpiece zero point

G56 ■ Additive shift
 ■ Programmed shift
 ■ Reference: Current workpiece zero point

G59 ■ Absolute shift
 ■ Programmed shift
 ■ Reference: Machine zero point



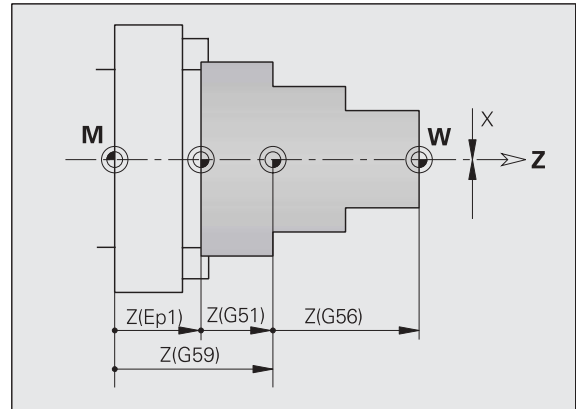
Additive zero offset G56

Shifts the workpiece zero point by "Z" (or "X"). The shift is given with respect to the currently active workpiece zero point.

If you shift the workpiece zero point several times with G56, the shift is always added to the currently active zero point.

Parameters

X, Z: Shift (X radius value) – default: 0



Absolute zero offset G59

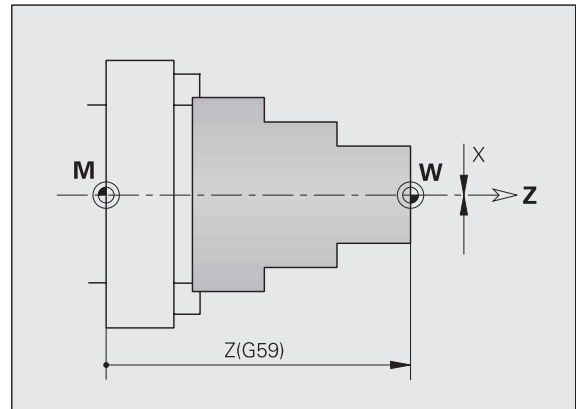
Sets the workpiece zero point to X, Z. The new zero point remains in effect to the end of the program.

Parameters

X, Z: Zero-point shift (X radius dimension)



G59 cancels all previous zero point displacements (with G51, G53..G55 or G59).



Mirror/shift contour G121

Mirrors and or shifts the workpiece blank contour and finished part contour. The contour is mirrored at the X axis and shifted in Z direction. The workpiece zero point is not affected.

G121 allows you to use the blank and finished part descriptions for front and rear-face machining.

Parameters

H: Mirroring – default: 0

- H=0: Contour shift – no mirroring
- H=1: Contour shift, mirroring, and reversal of the direction of contour description

Q: Mirroring the coordinate system (direction of the Z axis) - default: 0

- Q=0: No mirroring
- Q=1: Mirroring



- Lateral surface contours are mirrored/shifted like turning contours.
- Auxiliary contours are not mirrored.
- Remember when Q=1: the coordinate system and the contour are mirrored – H=1 mirrors only the contour.

Z: Shift – default: 0

D: Mirroring XC/XCR (mirroring/shifting the front and rear face contours) - default: 0

- D=0: No mirroring/shifting
- D=1: Mirroring/shifting

Continued ►

Example of rear side machining with opposing spindle.

■ Part transfer **with** mirroring of the coordinate system

...

N.. G121 H1 Q1 Z.. D1

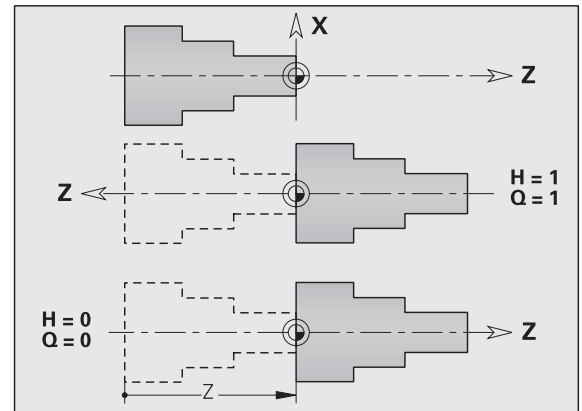
...

■ Part transfer **without** mirroring of the coordinate system.

...

N.. G121 H0 Q0 Z.. D1

...



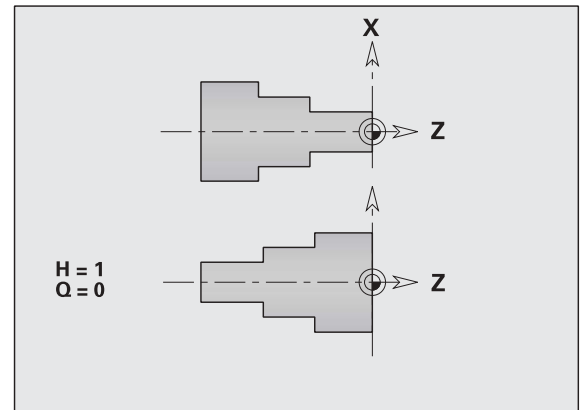
Example of rear side machining with a spindle

The workpiece is rechucked manually to machine the rear face.

...

N.. G121 H1 Q0 Z.. D1

...



4.6.7 Oversizes, Safety Clearances

Safety clearance G47

Safety clearance for the turning cycles: G810, G820, G830, G835, G860, G869, G890, the drilling cycles G71, G72, G74 and milling cycles G840...G846.

G47 without parameters activates the parameter values (machining parameters 2, ... – safety clearances).

Parameters

P: Safety clearance

Switch off oversize with G50

Switches off oversizes defined with G52/G39 Geo for the following cycle. G50 is programmed before the cycle.



G47 replaces safety clearance set in the machining parameters or that set in G147.

Switch off oversize G52

G52 has the same effect as G50! – Use G50.

Parameters

P: Oversize - is not evaluated

Safety clearance G147

Safety clearance for the milling cycles G840...G846 and drilling cycles G71, G72, G74.

Parameters

I: Safety clearance to the milling plane (only for milling operations)

K: Safety clearance in approach direction (feed)



G147 replaces safety clearance set in the machining parameters (machining parameters 2, ...) or that set in G47.

Axis-parallel oversize G57

G57 defines different oversizes for X and Z. Program G57 before the cycle call.

G57 is effective in the following cycles. After cycle run, the oversizes are

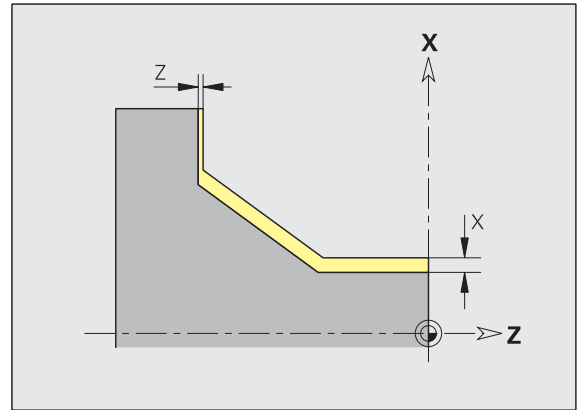
- deleted: G810, G820, G830, G835, G860, G869, G890
- **not** deleted: G81, G82, G83

Parameters

X, Z: Oversize (X diameter value) – only positive values



If the oversizes are programmed with G57 **and** in the cycle itself, the cycle oversizes apply.



Contour-parallel oversize (equidistant) G58

A negative oversize is permitted with G890. Program G58 before the cycle call.

G58 is effective in the following cycles. After cycle run, the oversizes are

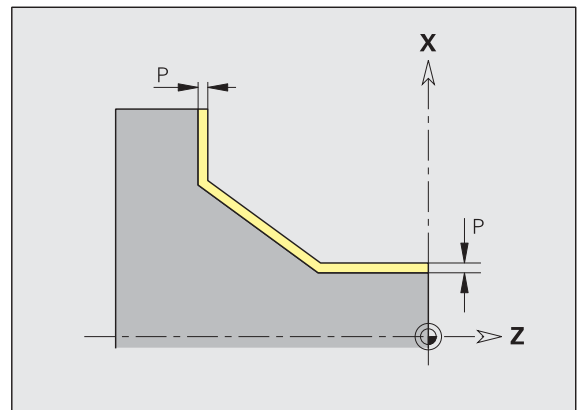
- deleted: G810, G820, G830, G835, G860, G869, G890
- **not** deleted: G83

Parameters

P: Oversize



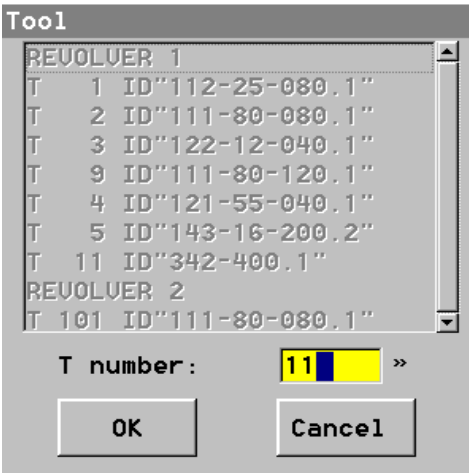
If an oversize is programmed with G58 **and** in the cycle, the oversize from the cycle is used.



4.6.8 Tools,Types of Compensation

Tool callT

The CNC PILOT displays the tool assignment defined in theTURRET section. You can enter theT number directly or select it from the tool list (switch with the CONTINUE soft key). See also "4.2.4Tool Programming."



(Changing the) cutter compensation G148

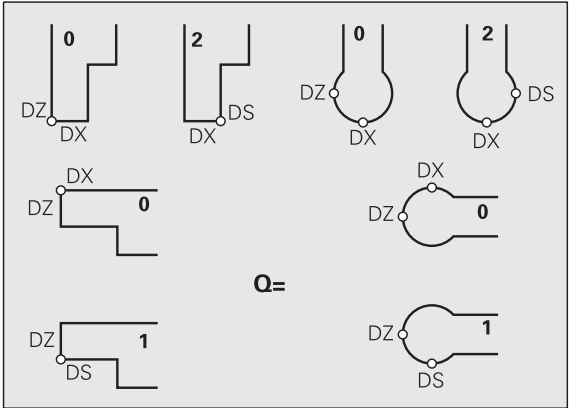
"O" defines the values compensating for wear. On program start and after a T command, DX, DZ are active.

Parameters

- O: Selection – default: 0
- O=0: DX, DZ active – DS inactive
 - O=1: DS, DZ active – DX inactive
 - O=2: DX, DS active – DZ inactive



The recessing cycles G860, G866, G869 automatically take the "correct" wear compensation into account.



Additive compensation G149

The CNC PILOT manages 16 tool-independent compensation values. A G149 followed by a "D number" activates the compensation – "G149 D900" switches the compensation off.

Parameters

- D: Additive compensation – default: D900; range: 900..916

Programming notes

- The compensation becomes effective after the tool has moved in the compensation direction by the compensation value. Therefore, program G149 one block before the block containing the path of traverse to which the compensation is to apply.
- An additive compensation remains effective until:
 - The next "G149 D900"
 - The next tool change
 - The end of the program

Example

...	
N.. G1 Z-25	
N.. G149 D901	[Activate the compensation]
N.. G1 X50	["Move" compensation: Position X50 + compensation]
N.. G1 Z-50	[Compensation is applied to contour element]
N.. G149 D900	[Deactivate compensation]
...	

Compensate right tool tip G150
Compensate left tool tip G151

Defines the tool reference point for recessing and button tools.

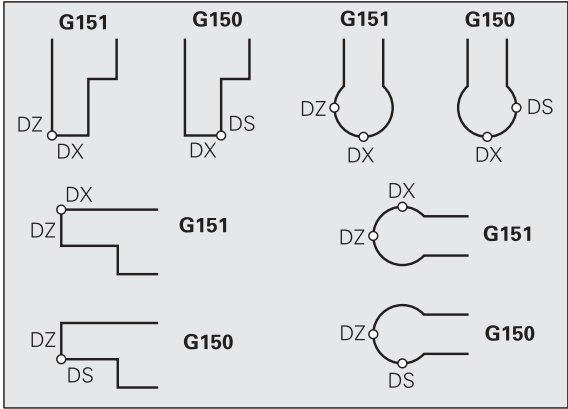
- G150: Reference point of the right tool tip
- G151: Reference point of the left tool tip

G150/G151 is effective from the block in which it is programmed and remains in effect up to

- The next tool change
- The end of the program



- The displayed actual values always refer to the tool tip defined in the tool data.
- If you use TRC, after G150/G151 you must also adjust G41/G42.



Linking tool dimensions G710

When aT command is programmed, the CNC PILOT replaces the previous tool dimensions with new tool dimensions. When you activate the adding function with "G710 Q1", the dimensions of the new tool are **added** to the dimensions of the previous tool.

Parameters

- Q: Add tool dimensions
- Q=0: Off
 - Q=1: On

Example for application

For full-surface machining, the workpiece is transferred to a rotating gripper after having been machined on the front face. The rear side is machined by stationary tools. To do this, the dimensions of the rotating gripper are added to the dimensions of the stationary tool.

. . .	Example of "adding tool dimensions"
REVOLVER 1 [TURRET]	
. . .	
T14 ID"SETUP PICKUP"	Rotating gripper
. . .	
REVOLVER 2 [TURRET]	Stationary tools on tool carrier 2
T2001 ID"116-80-080.1"	Roughing tool for rear-face machining
. . .	
BEARBEITUNG [MACHINING]	
. . .	
N100 T14	Insert rotating gripper
N101 L"EXGRIF" V1	Transfer workpiece from spindle to rotating gripper (expert program)
N102 G710 Q1	Add tool dimensions
N103 T2001	Add the dimensions of the rotating gripper and the stationary tool
. . .	

4.7 Turning Cycles

4.7.1 Contour-Based Turning Cycles

Finding the block references:

- ▶ Activate the contour graphics (GRAPHICS soft key)
- ▶ Set the cursor to NS/NE and press the CONTINUE soft key
- ▶ Using the horizontal arrow keys to select the contour element
- ▶ The vertical arrow keys can be used to switch between contours (also face contours, etc.)
- ▶ Confirm the block of the contour element with ENTER

Longitudinal roughing G810

G810 machines the contour area defined by "NS, NE." The CNC PILOT uses the tool definition to distinguish between external and internal machining. With "NS – NE" you specify the machining direction.

If the contour to be machined consists of one element, then:

- If you program only NS: Machining in contour definition direction
- If you program NS and NE: Machining against the contour definition direction

If required, the area to be machined is divided into several sections (example: with contour valleys).

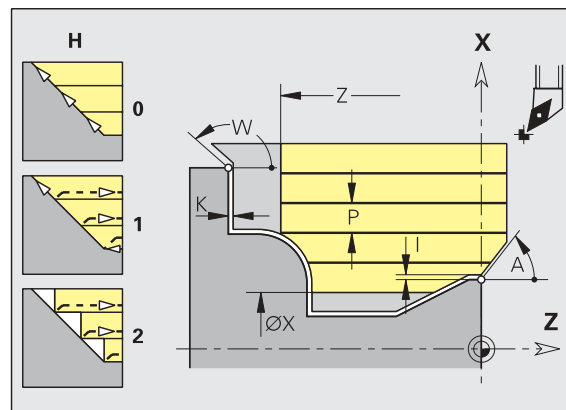
The simplest way of programming is specifying NS, NE and P.

Parameters

- NS: Starting block number (beginning of contour section)
 NE: End block number (end of contour section)
 P: Maximum infeed
 I: Oversize in X direction (diameter value)—default: 0
 K: Oversize in Z direction—default: 0
 E: Approach behavior
 ■ E=0: Descending contours are not machined
 ■ E>0: Approach behavior
 ■ No input: Feed rate reduced depending on approach angle—maximum reduction: 50%
 X: Cutting limit in X direction (diameter value)—default: no cutting limit
 Z: Cutting limit in Z direction—default: no cutting limit
 H: Type of contour smoothing—default:
 ■ H=0: smoothing after each cut
 ■ H=1: lift off at under 45°, smoothing after last cut
 ■ H=2: lift off at under 45°, no smoothing



If you press the vertical arrow keys, the CNC PILOT also considers contours that are not displayed on the screen.



Cycle run

- 1 Calculate the areas to be machined and the cutting segmentation (infeeds).
- 2 Approach workpiece for first pass from starting point, taking the safety clearance into account (first in Z direction, then in X).
- 3 Move at feed rate to target point Z.
- 4 Depending on H:
 - H=0: Cut along the contour
 - H=1 or 2: Retract at 45°
- 5 Return at rapid traverse and approach for next pass.
- 6 Repeat 3 to 5 until target point X has been reached.
- 7 If required, repeat 2 to 6 until all areas have been machined.
- 8 H=1: Smoothen contour.
- 9 Retract according to "Q."

Continued▶

- A: Approach angle (reference: Z axis)—default: 0°/180° (parallel to Z-axis)
- W: Departing angle (reference: Z axis)—default: 90°/270° (perpendicular to Z axis)
- Q: Type of retraction after machining—default: 0
 - Q=0: Return to starting point (first in X direction, then in Z)
 - Q=1: Position in front of finished contour
 - Q=2: Move to clearance height and stop
- V: Identifier beginning/end—default: 0

A chamfer/rounding arc is being machined:

 - V=0: At beginning and end
 - V=1: At beginning
 - V=2: At end
 - V=3: No machining
 - V=4: Chamfer/rounding is being machined—not the basic element (prerequisite: Contour section with an element)
- D: Omit element (influences the machining of undercuts, relief turns: see table)—default: 0
- B: Slide lead for 4-axis machining
 - B=0: Both slides work on the same diameter—with double feed rate
 - B<>0: Distance to “leading” slide (the lead). The slides work on different diameters with the same feed rate.
 - B<0: The slide with larger number leads.
 - B>0: The slide with smaller number leads.



Cutting limitation: The tool position before the cycle call determines the effect of a cutting limit. The CNC PILOT machines the area to the right or to the left of the cutting limit, depending on which side the tool has been positioned before the cycle is called.

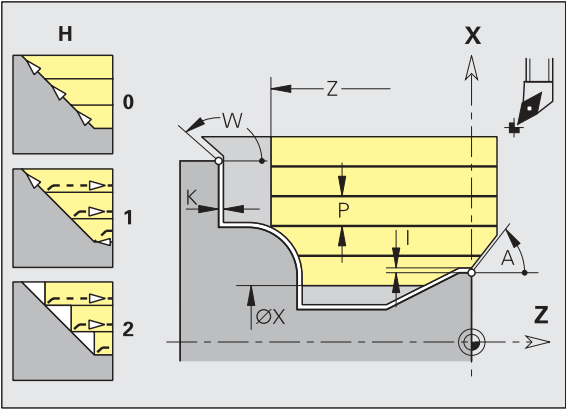
Cutter radius compensation: Active

G57 oversize: “Enlarges” the contour (also inside contours)

G58 oversize:

- >0: “enlarges” the contour
- <0: is not considered

G57/G58 oversizes are deleted after cycle end



4 axis operation

- When working on the same diameter, both slides start simultaneously.
- When working on different diameters, the second slide starts when the leading slide has reached “lead B.” This is synchronized at every step.

Each slide advances by the calculated depth of cut. If the slides do not have to execute the same number of cuts, the leading slide executes the last cut.

With “constant cutting speed,” the cutting speed depends on the speed of the leading slide. The leading tool does not retract until the subsequent tool is ready for use.



On **4 axis cycles**, ensure that the tools are identical (tool type, cutting edge radius, cutting edge angle, etc.).

D	G22	G23	G23	G25	G25	G25
=	H0	H1	H4	H5/6	H7.9	
0	•	•	•	•	•	•
1	•	•	•	–	–	–
2	•	•	–	•	•	•
3	•	•	–	–	–	–
4	•	•	–	•	•	–

“•”: Skip elements

Face roughing G820

G820 machines the contour area defined by “NS, NE.” The CNC PILOT uses the tool definition to distinguish between external and internal machining. With “NS – NE” you specify the machining direction.

If the contour to be machined consists of one element, then:

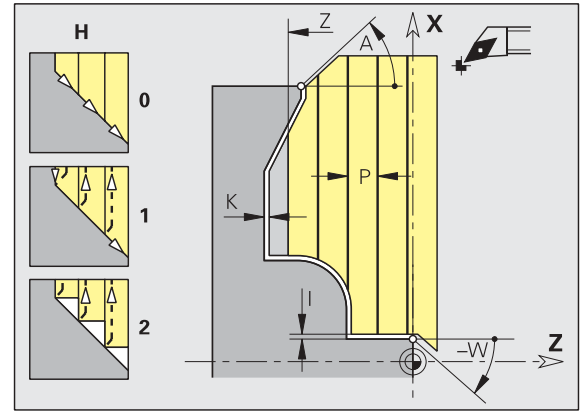
- If you program only NS, machining is in contour definition direction
- If you program NS and NE, machining is against the contour definition direction

If required, the area to be machined is divided into several sections, for example, for machining contour valleys.

The simplest way of programming is specifying NS, NE and P.

Parameters

- NS: Starting block number (beginning of contour section)
 NE: End block number (end of contour section)
 P: Maximum infeed
 I: Oversize in X direction (diameter value)—default: 0
 K: Oversize in Z direction—default: 0
 E: Approach behavior
 ■ E=0: Descending contours are not machined
 ■ E>0: Approach behavior
 ■ No input: Feed rate reduced depending on approach angle—maximum reduction: 50%
 X: Cutting limit in X direction (diameter value)—default: no cutting limit
 Z: Cutting limit in Z direction—default: no cutting limit
 H: Type of contour smoothing—default:
 ■ H=0: smoothing after each cut
 ■ H=1: lift off at under 45°, smoothing after last cut
 ■ H=2: lift off at under 45°, no smoothing
 A: Approaching angle (reference: Z axis)—default: 90°/270° (perpendicular to Z-axis)
 W: Departing angle (reference: Z axis)—default: 0°/180° (parallel to Z-axis)
 Q: Type of retraction after machining—default: 0
 ■ Q=0: Return to starting point (first in X direction, then in Z)
 ■ Q=1: Position in front of finished contour
 ■ Q=2: Move to clearance height and stop
 V: Identifier beginning/end—default: 0
 A chamfer/rounding arc is being machined:
 ■ V=0: At beginning and end
 ■ V=1: At beginning
 ■ V=2: At end



Cycle run

- 1 Calculate the areas to be machined and the cutting segmentation (infeeds).
- 2 Approach workpiece for first pass from starting point, taking the safety clearance into account (first in X direction, then in Z).
- 3 Move at feed rate to target point X.
- 4 Depending on H:
 ■ H=0: Cut along the contour
 ■ H=1 or 2: Retract at 45°
- 5 Return at rapid and approach for next pass.
- 6 Repeat 3 to 5 until target point Z has been reached.
- 7 If required, repeat 2 to 6 until all areas have been machined.
- 8 H=1: Smoothen contour.
- 9 Retract according to “Q.”



Cutting limitation: The tool position before the cycle call determines the effect of a cutting limit. The CNC PILOT machines the area to the right or to the left of the cutting limit, depending on which side the tool has been positioned before the cycle is called.

Cutter radius compensation: Active

G57 oversize: “Enlarges” the contour (also inside contours)

G58 oversize:

- >0: “enlarges” the contour
- <0: is not considered

G57/G58 oversizes are deleted after cycle end

Continued ►


- V=3: No machining
 - V=4: Chamfer/rounding is being machined—not the basic element (prerequisite: Contour section with an element)
- D: Omit element (influences the machining of undercuts, relief turns: see table)—default: 0
- B: Slide lead for 4-axis machining
- B=0: Both slides work on the same diameter—with double feed rate
 - B<>0: Distance to “leading” slide (the lead). The slides work on different diameters with the same feed rate.
 - B<0: The slide with larger number leads.
 - B>0: The slide with smaller number leads.

4 axis operation

- When working on the same diameter, both slides start simultaneously.
 - When working on different diameters, the second slide starts when the leading slide has reached “lead B.” This is synchronized at every step. Each slide advances by the calculated depth of cut.
- If the slides do not have to execute the same number of cuts, the leading slide executes the last cut.
- With “constant cutting speed,” the cutting speed depends on the speed of the leading slide. The leading tool does not retract until the subsequent tool is ready for use.

D	G22	G23	G23	G25	G25	G25
=	H0	H1	H4	H5/6	H7..9	
0	•	•	•	•	•	•
1	•	•	•	–	–	–
2	•	•	–	•	•	•
3	•	•	–	–	–	–
4	•	•	–	•	•	–

“•”: Skip elements



On **4 axis cycles**, ensure that the tools are identical (tool type, cutting edge radius, cutting edge angle, etc.).

Contour parallel roughing G830

G830 machines the contour area defined by "NS, NE" parallel to the contour. The CNC PILOT uses the tool definition to distinguish between external and internal machining. With "NS – NE" you specify the machining direction.

If the contour to be machined consists of one element, then:

- If you program only NS, machining is in contour def. direction
- If you program NS and NE, machining is against the contour definition direction

If required, the area to be machined is divided into several sections, for example, for machining contour valleys.

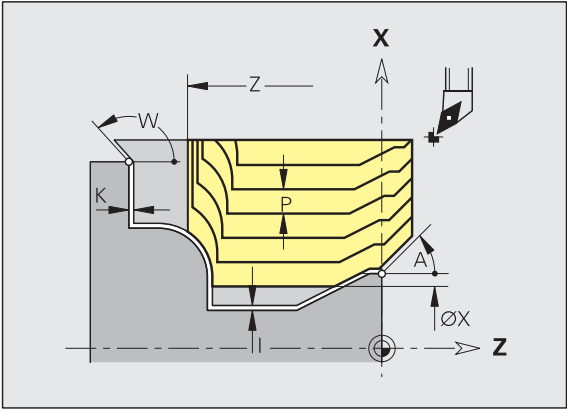
The simplest way of programming is specifying NS, NE and P.

Parameters

- NS: Starting block number (beginning of contour section)
NE: End block number (end of contour section)
P: Maximum infeed
I: Oversize in X direction (diameter value)—default: 0
K: Oversize in Z direction—default: 0
X: Cutting limit in X direction (diameter value)—default: none
Z: Cutting limit in Z direction—default: no cutting limit
A: Approach angle (reference: Z axis)—default: 0°/180° (parallel to Z-axis)
W: Departing angle (reference: Z axis)—default: 90°/270° (perpendicular to Z-axis)
Q: Type of retraction after machining—default: 0
 ■ Q=0: Return to starting point (first in X direction, then in Z)
 ■ Q=1: Position in front of finished contour
 ■ Q=2: Move to clearance height and stop
V: Identifier beginning/end—default: 0
 A chamfer/rounding arc is being machined:
 ■ V=0: At beginning and end
 ■ V=1: At beginning
 ■ V=2: At end
 ■ V=3: No machining
 ■ V=4: Chamfer/rounding is being machined—not the basic element (prerequisite: Contour section with an element)
D: Omit element (influences the machining of undercuts, relief turns: see table)—default: 0

D	G22	G23	G23	G25	G25	G25
=	H0	H1	H4	H5/6	H7.9	
0	•	•	•	•	•	•
1	•	•	•	–	–	–
2	•	•	–	•	•	•
3	•	•	–	–	–	–
4	•	•	–	•	•	–

"•": Skip elements



Cycle run

- 1 Calculate the areas to be machined and the cutting segmentation (infeeds).
- 2 Approach workpiece for first pass from starting point, taking the safety clearance into account.
- 3 Execute the first cut (roughing).
- 4 Return at rapid traverse and approach for next pass.
- 5 Repeat 3 to 4 until the complete area has been machined.
- 6 If required, repeat 2 to 5 until all areas have been machined.
- 7 Retract as programmed in "Q."



Cutting limitation: The tool position before the cycle call determines the effect of a cutting limit. The CNC PILOT machines the area to the right or to the left of the cutting limit, depending on which side the tool has been positioned before the cycle is called.

Cutter radius compensation: Active

G57 oversize: "Enlarges" the contour (also inside contours)

G58 oversize:
■ >0: "enlarges" the contour
■ <0: is not considered

G57/G58 oversizes are deleted after cycle end

Contour-parallel with neutral tool G835

G830 machines the contour area defined by “NS, NE” parallel to the contour and bidirectionally. The CNC PILOT uses the tool definition to distinguish between external and internal machining.

If required, the area to be machined is divided into several sections, for example, for machining contour valleys.

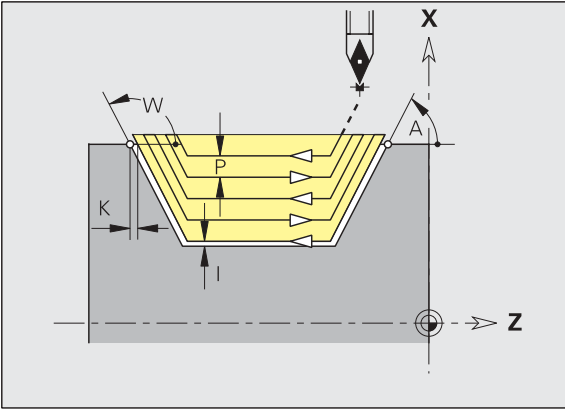
The simplest way of programming is specifying NS, NE and P.

Parameters

- NS: Starting block number (beginning of contour section)
- NE: End block number (end of contour section)
- P: Maximum infeed
- I: Oversize in X direction (diameter value)—default: 0
- K: Oversize in Z direction—default: 0
- X: Cutting limit in X direction (diameter value)—default: no cutting limit
- Z: Cutting limit in Z direction—default: no cutting limit
- A: Approach angle (reference: Z axis)—default: 0°/180° (parallel to Z-axis)
- W: Departing angle (reference: Z axis)—default: 90°/270° (perpendicular to Z-axis)
- Q: Type of retraction after machining—default: 0
- Q=0: Return to starting point (first in X direction, then in Z)
 - Q=1: Position in front of finished contour
 - Q=2: Move to clearance height and stop
- V: Identifier beginning/end—default: 0
- A chamfer/rounding arc is being machined:
- V=0: At beginning and end
 - V=1: At beginning
 - V=2: At end
 - V=3: No machining
 - V=4: Chamfer/rounding is being machined—not the basic element (prerequisite: Contour section with an element)
- D: Omit element (influences the machining of undercuts, relief turns: see table)—default: 0

D	G22	G23	G23	G25	G25	G25
=	H0	H1	H4	H5/6	H7..9	
0	•	•	•	•	•	•
1	•	•	•	–	–	–
2	•	•	–	•	•	•
3	•	•	–	–	–	–
4	•	•	–	•	•	–

“•”: Skip elements



Cycle run

- 1 Calculate the areas to be machined and the cutting segmentation (infeeds).
- 2 Approach workpiece for first pass from starting point, taking the safety clearance into account.
- 3 Execute the first cut (roughing).
- 4 Approach for the next pass and execute the next cut (roughing) in the opposite direction.
- 5 Repeat 3 to 4 until the complete area has been machined.
- 6 If required, repeat 2 to 5 until all areas have been machined.
- 7 Retract as programmed in “Q.”



Cutting limitation: The tool position before the cycle call determines the effect of a cutting limit. The CNC PILOT machines the area to the right or to the left of the cutting limit, depending on which side the tool has been positioned before the cycle is called.

Cutter radius compensation: Active

G57 oversize: “Enlarges” the contour (also inside contours)

G58 oversize:

- >0: “enlarges” the contour
- <0: is not considered

G57/G58 oversizes are deleted after cycle end

Recessing G860

G860 machines (indents) the contour area defined by “NS, NE” axially/radially. The contour to be machined may contain various valleys. The CNC PILOT uses the tool definition to distinguish between external and internal machining, or between radial and axial recesses.

Calculation of the cut segmentation (SBF: see machining parameter 6): maximum offset = $SBF * \text{width of cut}$

With “NS – NE” you specify the machining direction. If the contour to be machined consists of one element, then:

- If you program only NS: Machining in contour definition direction
- If you program NS and NE: Machining against the contour definition direction

If required, the area to be machined is divided into several sections, for example, for machining contour valleys.

The simplest way of programming is specifying NS, or NS and NE.

Parameters

NS: Starting block number (beginning of contour section, or reference to recess defined by G22/G23-Geo).

NE: End block number (end of contour section)—omit for contour defined by G22/G23-Geo.

I: Oversize in X direction (diameter value)—default: 0

K: Oversize in Z direction—default: 0

Q: Sequence—default: 0

- Q=0: Roughing and finishing
- Q=1: Roughing only
- Q=2: Finishing only

X: Cutting limit in X direction (diameter value)—default: no cutting limit

Z: Cutting limit in Z direction—default: no cutting limit

Code start/end—default: 0

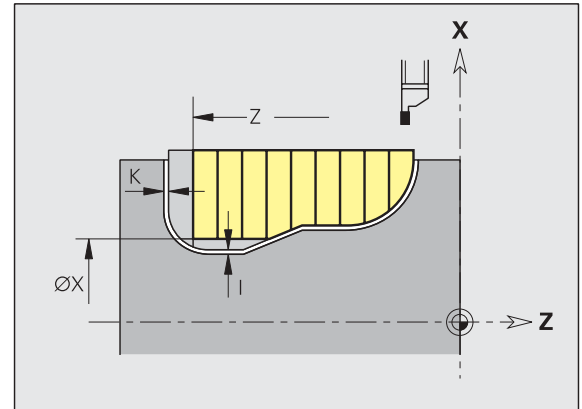
A chamfer/rounding arc is machined:

- V=0: At the start and end
- V=1: At the start
- V=2: At end
- V=3: No machining

E: Feed rate for finishing—default: Active feed rate

H: Retraction at end of cycle—default: 0

- H=0: Return to starting point (axial recess: first Z and then X direction; radial recess: first X and then Z direction)
- H=1: Position in front of the finished contour
- H=2: Move to clearance height and stop



Cycle run (where Q=0 or 1)

- 1 Calculate the areas to be machined and the cutting segmentation.
- 2 Approach workpiece for first pass from starting point, taking the safety clearance into account (radial recess: first in Z, then in X direction; axial recess: first in X, then in Z direction)
- 3 Execute first cut (roughing).
- 4 Return at rapid traverse and approach for next pass.
- 5 Repeat 3 to 4 until the complete area has been machined.
- 6 If required, repeat 2 to 5 until all areas have been machined.
- 7 Q=0: Finish-machine the contour.



Cutting limitation: The tool position before the cycle call determines the effect of a cutting limit. The CNC PILOT machines the area to the right or to the left of the cutting limit, depending on which side the tool has been positioned before the cycle is called.

Cutter radius compensation: Active

G57 oversize: “Enlarges” the contour (also inside contours)

G58 oversize:

- >0: “enlarges” the contour
- <0: is not considered

G57/G58 oversizes are deleted after cycle end

Recessing cycle G866

G866 generates a recess defined by G22-Geo. The CNC PILOT uses the tool definition to distinguish between external and internal machining, or between radial and axial recesses.

Calculation of the cut segmentation (SBF: see machining parameter 6): maximum offset = $SBF * \text{width of cut}$

Parameters

NS: Block number (reference to G22-Geo)

I: Allowance (with precutting)—default: 0

■ I=0: Recess is machined in one run

■ I>0: Precutting is carried out in the first run; and finishing in the second

E: Dwell time—no input: Corresponds to a spindle revolution

■ With I=0: For each recess

■ With I>0: Only during finishing



Cutter radius compensation is active.

Oversizes are not taken into account.

Cycle run

1 Calculate the proportioning of cuts.

2 Approach workpiece for first pass from starting point (radial recess: first in Z, then in X direction; axial recess, first in X, then in Z direction).

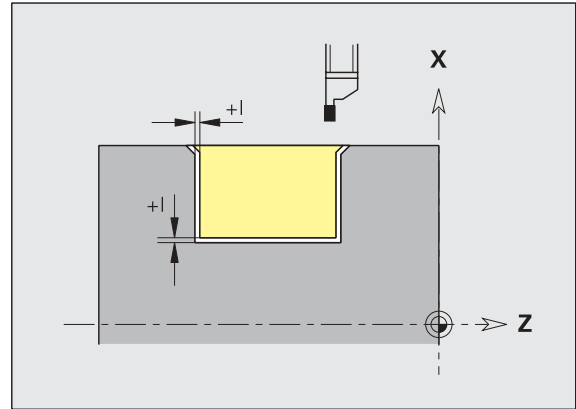
3 Execute the first cut according to "I."

4 Return at rapid traverse and approach for next pass.

5 I=0: "E" period of dwell.

6 Repeat 3 to 4 until the recess has been machined.

7 I>0: Finish-machine the contour.



Recess turning cycle G869

G869 machines (indents) the contour area defined by "NS, NE" axially/radially. The workpiece is machined by alternate recessing and roughing movements. The machining process requires a minimum of retraction and infeed movements.

The contour to be machined may contain various valleys. If required, the area to be machined is divided into several sections.

The CNC PILOT uses the tool definition to distinguish between radial and axial recesses.

With "NS – NE" you specify the machining direction. If the contour to be machined consists of one element, then:

- If you program only NS: Machining in contour definition direction
- If you program NS and NE: Machining against the contour definition direction

Depending on factors such as workpiece material or feed rate, the tool tip is displaced during a turning operation. You can correct the resulting infeed error with "turning depth compensation factor" R. The value is usually determined empirically.

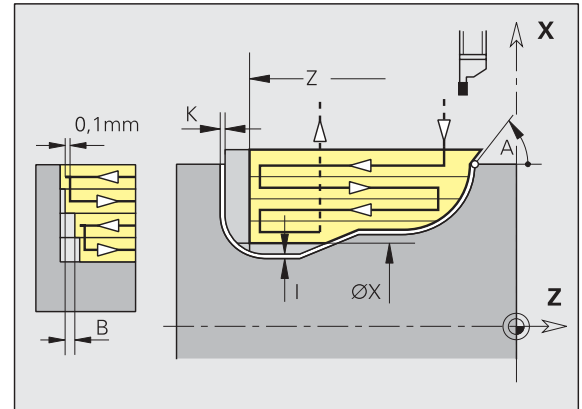
After the second infeed movement, during the transition from turning to recessing, the path to be machined is reduced by "Offset width B." Each time the system switches on this side, the path is reduced by "B"—in addition to the previous offset. The total offset is limited to 80% of the effective cutting width (effective cutting width = cutting width – 2 * cutting radius). If required, the CNC PILOT reduces the programmed offset width. After precutting, the remaining material is removed with a single cut.

Unidirectional turning (U=1): Roughing is in the machining direction "NS – NE."

The simplest way of programming is specifying NS or NS, NE and P.

Parameters

- NS: Starting block number (beginning of contour section, or reference to G22/G23 Geo recess)
- NE: End block number (end of contour section)—omit for contour defined by G22/G23-Geo.
- P: Maximum infeed
- R: Turning depth compensation for finishing—default: 0
- I: Oversize in X direction (diameter value)—default: 0
- K: Oversize in Z direction—default: 0
- X: Cutting limit in (diameter value)—default: no cutting limit
- Z: Cutting limit—default: no cutting limit



Cycle run (where Q=0 or 1)

- 1 Calculate the areas to be machined and the cutting segmentation.
- 2 Approach workpiece for first pass from starting point, taking the safety clearance into account (radial recess: first in Z, then in X direction; axial recess: first in X, then in Z direction)
- 3 Execute the first cut (recessing).
- 4 Machine perpendicularly to recessing direction (turning).
- 5 Repeat 3 to 4 until the complete area has been machined.
- 6 If required, repeat 2 to 5 until all areas have been machined.
- 7 Q=0: Finish-machine the contour.



G869 requires **tools** of the type 26*.

Cutting limitation: The tool position before the cycle call determines the effect of a cutting limit. The CNC PILOT machines the area to the right or to the left of the cutting limit, depending on which side the tool has been positioned before the cycle is called.

Cutter radius compensation: Active

G57 oversize: "Enlarges" the contour (also inside contours)

G58 oversize:

- >0: "enlarges" the contour
- <0: is not considered

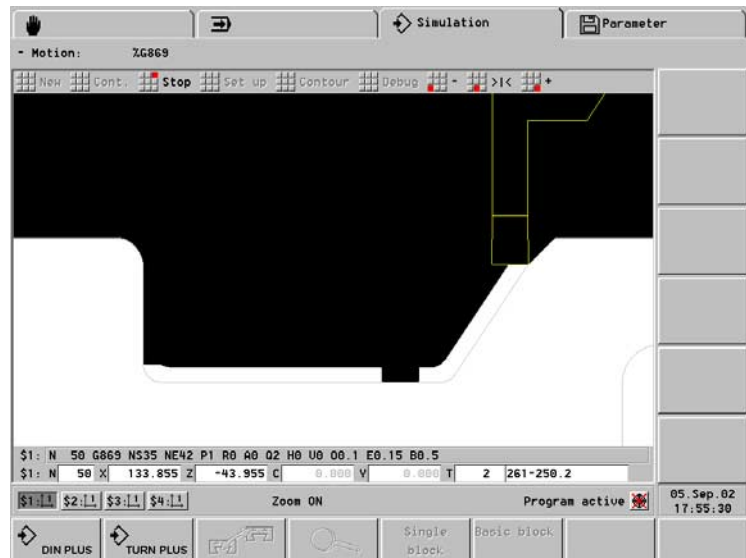
G57/G58 oversizes are deleted after cycle end

Continued ►

- A, W: Approach angle, departure angle—default: opposite from the recessing direction
- Q: Sequence—default: 0
- Q=0: Roughing and finishing
 - Q=1: Roughing only
 - Q=2: Finishing only
- U: Unidirectional turning—default: 0
- U=0: bidirectional
 - U=1: unidirectional in contour direction
- H: Retraction at end of cycle—default: 0
- H=0: Return to starting point (axial recess: first Z and then X direction; radial recess: first X and then Z direction)
 - H=1: Position in front of the finished contour
 - H=2: Move to clearance height and stop
- Code start/end—default: 0
- A chamfer/rounding arc is machined:
- V=0: At the start and end
 - V=1: At the start
 - V=2: At end
 - V=3: No machining
- O: Recessing feed rate—default: Active feed rate
- E: Feed rate for finishing—default: Active feed rate
- B: Offset width—default: 0

Machining information

- **Transition from turning to recessing:** Before the transition from turning to recessing, the CNC PILOT retracts the tool by 0.1 mm. Thus an offset cutting edge is adjusted for the recessing operation, independent of "offset width B."
- **Inside radii and chamfers:** Depending on the recessing width and the radii of rounding, single cuts preventing a "fluid transition" from recessing to turning are executed before the rounding is machined. This prevents damage to the tool.
- **Edges:** Edges are recessed. This prevents "residual rings."



Finish contour G890

G890 finishes the contour area defined by „NS, NE“ parallel to the contour in one pass and takes chamfers/rounding into account. Undercuts are machined where tool geometry permits.

The CNC PILOT uses the tool definition to distinguish between external and internal machining.

With „NS – NE“ you specify the machining direction. If the contour to be machined consists of one element, the following applies:

- The contour is machined in the defined direction if you program only NS
- The contour is machined opposite to the defined direction if you program NS and NE.

You activate the **residual finishing** with Q=4 (example: hollowing with finishing tools that machine in the direction opposite to that defined). The CNC PILOT knows the areas that have already been machined and does not machine them again. If Q=4, you cannot influence the approach type. It is determined by the finishing cycle.

Note for **small chamfers/rounding**:

- Peak-to-valley height or feed rate (with G95-Geo) are not programmed: The CNC PILOT automatically reduces the feed rate. The chamfer/rounding arc is machined with at least 3 revolutions.
- If peak-to-valley height or feed rate (with G95 Geo) are programmed: no automatic feed reduction

For chamfers/rounding which, as a result of their size, are machined with at least three revolutions, the feed rate is not reduced automatically.

Parameters

NS: Starting block number (beginning of contour section)

NE: End block number (end of contour section)

E: Approach behavior

- E=0: Descending contours are not machined
- E>0: Approach behavior
- No input: Feed rate reduced depending on approach angle —maximum reduction: 50%

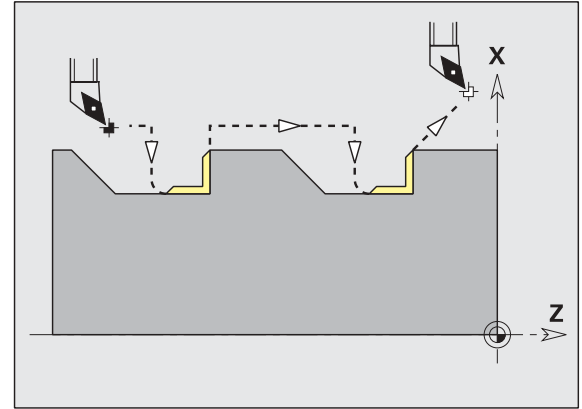
V: Identifier beginning/end—default: 0

A chamfer/rounding arc is being machined:

- V=0: At beginning and end
- V=1: At beginning
- V=2: At end
- V=3: No machining
- V=4: Chamfer/rounding is being machined—not the basic element (prerequisite: Contour section with an element)

Type of approach—default: 0

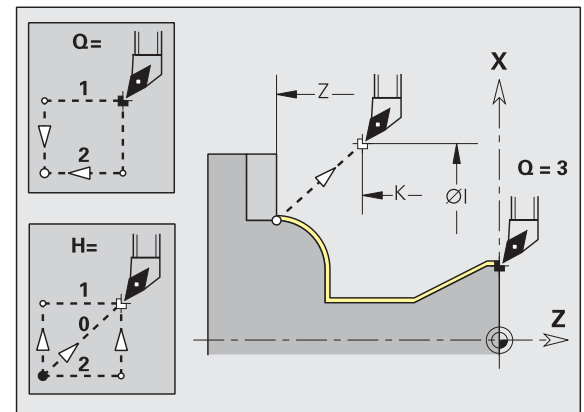
- Q=0: Automatic selection—the CNC PILOT checks:
 - Diagonal approach
 - First X, then Z direction



G890 Q4—Residual finishing



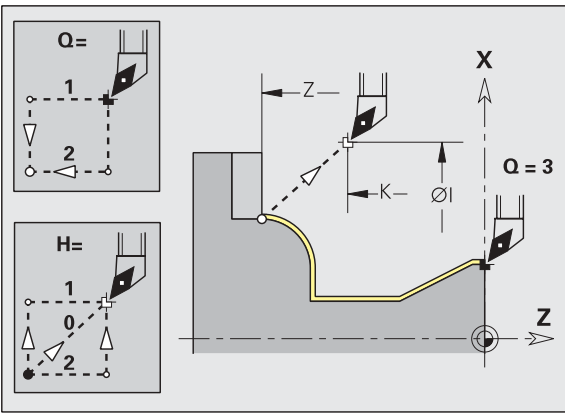
During **residual finishing** (G890 – Q4), the CNC PILOT checks whether the tool can move into the contour valley without a collision. The collision check is based on tool parameter „width dn“ (see “8.1.2 Notes on Tool Data”).



Continued ►

- Equidistant around the barrier
- Omission of the first contour element if the start position is inaccessible
 - Q=1: First X, then Z direction
 - Q=2: First Z, then X direction
 - Q=3: No approach—tool is in the proximity of the starting point
 - Q=4: Residual finishing

- H: Retraction—default: 3
Tool lifts off under 45° in the opposite direction to machining and moves as follows to the position “I, K”:
- H=0: Diagonal
 - H=1: First X, then Z direction
 - H=2: First Z, then X direction
 - H=3: Remains at safety clearance
 - H=4: No retraction movement—tool remains at the end coordinate
- Z: Cutting limit (diameter)—default: no cutting limit
- Z: Cutting limit—default: no cutting limit
- D: Omit element (influences the machining of undercuts, relief turns, and recessing: see table)—default: 1
- I, K: End point that is appropriate at the end of the cycle (I diameter value)
- O: Feed rate reduction—default: 0
 - O=0: No feed rate reduction
 - O=1: Feed rate reduction active



Cutting limitation: The tool position before the cycle call determines the effect of a cutting limit. The CNC PILOT machines the area to the right or to the left of the cutting limit, depending on which side the tool has been positioned before the cycle is called.

G57 oversize: “enlarge” the contour (also inside contours)

G58 oversize:

- >0: “enlarges” the contour
- <0: “reduces” the contour

G57/G58 oversizes are deleted after cycle end

Undercuts/undercut combinations can be omitted as follows:

D	G22	G23	G23	G25	G25	G25
=	H0	H1	H4	H5/6	H7.9	K
0	•	•	•	•	•	•
1	•	•	–	•	–	–
2	•	•	–	•	•	•
3	•	•	•	•	–	–
4	•	•	–	•	•	–
5	•	•	–	•	–	–
6	•	•	–	•	–	•
7	–	–	–	–	–	–

“•”: Skip elements

Other D codes for skipping undercuts/recesses. Add the codes if you want to skip several undercuts/recesses:

G call	Function	D code
G22	Recess for sealing ring	512
G22	Recess for guard ring	1.024
G23 H0	General recess	256
G23 H1	Relief turn	2.048
G23 H4	Undercut type U	32.768
G23 H5	Undercut type E	65.536
G23 H6	Undercut type F	131.072
G23 H7	Undercut type G	262.144
G23 H8	Undercut type H	524.288
G23 H9	Undercut type K	1.048.576

4.7.2 SimpleTurning Cycles

End of cycle G80

Concludes the fixed cycles.

Simple longitudinal roughing G81

G81 machines (roughs) the contour area described by the current tool position and "X, Z." If you wish to machine an oblique cut, you can define the angle with I and K.

The CNC PILOT uses the position of the target point to distinguish between external and internal machining.

The proportioning of cuts is calculated so that an "abrasive cut" is avoided and the calculated infeed distance \leq maximum infeed I.

Oversizes:

- G57 oversizes
 - are calculated with algebraic sign (oversizes are therefore impossible for inside contour machining)
 - stay in effect after cycle end
- G58 oversizes: are not taken into account

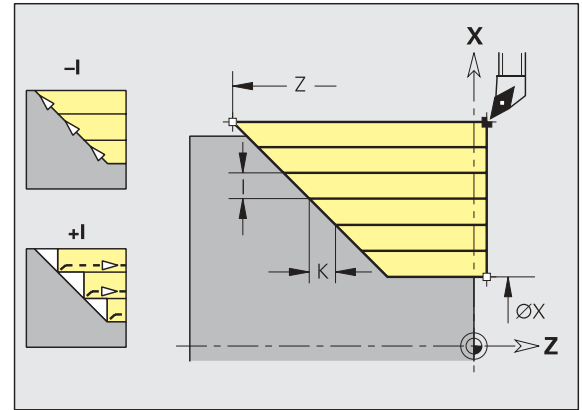
Cycle run

- 1 Calculate the cut segmentation (infeeds).
- 2 Approach workpiece for first pass from starting point on paraxial path.
- 3 Move at feed rate to target point Z.
- 4 Depending on the algebraic sign of I:
 - $I < 0$: Machine contour outline.
 - $I > 0$: Retract by 1 mm at 45° .
- 5 Return at rapid traverse and approach for next pass.
- 6 Repeat 3 to 5 until target point X has been reached.
- 7 Move to:
 - X – last position at which the tool retracts.
 - Z – starting point of cycle.

Parameters

X/Z: Contour target point (X diameter)

- I: Maximum infeed distance in X direction:
 - $I < 0$: With machining contour outline.
 - $I > 0$: Without machining contour outline.
- K: Offset in Z direction – default: 0
- Q: G function Infeed – default: 0
 - 0: Infeed with G0 (rapid traverse).
 - 1: Infeed with G1 (feed rate).



■ **Programming X, Z:** absolute, incremental or modal

■ **Cutter radius compensation:** Not active

■ **Safety clearance** After every step: 1 mm.

Simple face roughing G82

G82 machines (roughs) the contour area defined by the current tool position and "X, Z." If you wish to machine an oblique cut, you can define the angle with I and K.

The CNC PILOT uses the position of the target point to distinguish between external and internal machining.

The proportioning of cuts is calculated so that an "abrasive cut" is avoided and the calculated infeed distance \leq maximum infeed K.

Oversizes:

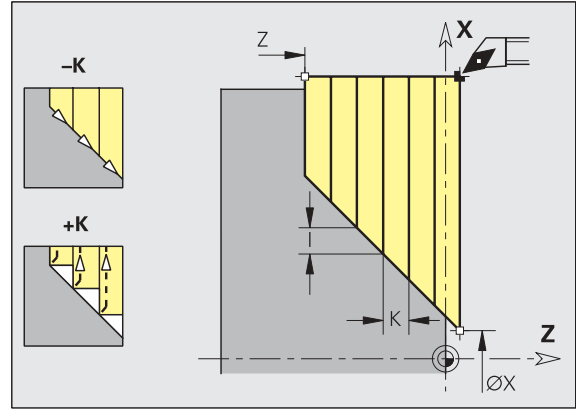
- G57 oversizes
 - are calculated with algebraic sign (oversizes are therefore impossible for inside contour machining)
 - stay in effect after cycle end
- G58 oversizes: are not taken into account

Cycle run

- 1 Calculate the cut segmentation (infeeds).
- 2 Approach workpiece for first pass from starting point on paraxial path.
- 3 Move at feed rate to target point X.
- 4 Depending on algebraic sign of K:
 - $K < 0$: Machine contour outline.
 - $K > 0$: Retract by 1 mm at 45°.
- 5 Return at rapid traverse and approach for next pass.
- 6 Repeat 3 to 5 until target point Z has been reached.
- 7 Move to:
 - X – starting point of cycle.
 - Z – last position at which the tool retracts.

Parameters

- X/Z: Contour target point (X diameter)
- I: Offset in X direction – default: 0
- K: Maximum infeed distance
 - $K < 0$: With machining contour outline.
 - $K > 0$: Without machining contour outline.
- Q: G function Infeed – default: 0
 - 0: Infeed with G0 (rapid traverse).
 - 1: Infeed with G1 (feed rate).



- **Programming X, Z:** absolute, incremental or modal
- **Tool radius compensation:** Not active
- **Safety clearance** After every step: 1 mm.

Simple contour repeat cycle G83

G83 carries out the functions programmed in the following blocks (simple traverses or cycles without a contour definition) more than once. G80 ends the machining cycle.

If the number of infeeds differs for the X and Z axes, the tool first advances in both axes with the programmed values. The infeed is set to zero if the target value for one direction is reached.

Note on programming G83

- G83 must be alone in a block.
- G83 must not be programmed with K variables.
- G83 must not be nested, not even by calling subprograms.

Oversizes:

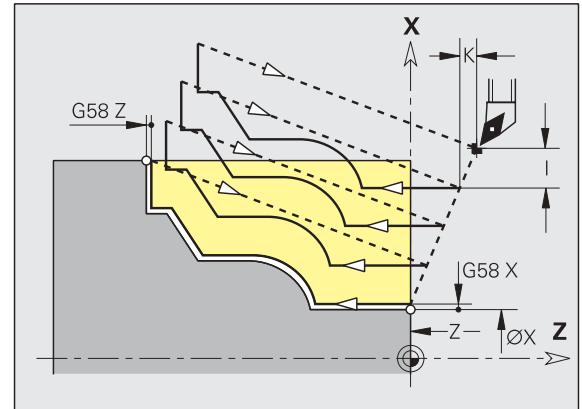
- G57 oversizes
 - are calculated with algebraic sign (oversizes are therefore impossible for inside contour machining)
- G58 oversizes: are taken into account if you are working with TRC
- G57 and G58 oversizes stay in effect after cycle end

Cycle run

- 1 Start the cycle execution from the current tool position.
- 2 Advance by the infeed distance defined in I and K.
- 3 Execute the machining operation which is programmed in the blocks after G83, taking the distance from the tool position to the contour start point as an "oversize."
- 4 Return on a diagonal path.
- 5 Repeat 2 to 4 until the contour target point has been reached.
- 6 Return to the starting point of the cycle.

Parameters

- X/Z: Contour target point (X diameter) - default: Transfer the last X/Z coordinate
- Il: Maximum infeed in X direction (radius) – default: 0
- K: Maximum infeed in Z direction – default: 0



■ **Cutter radius compensation:** Not active. – You can program the TRC separately with G40..G42.

■ **Safety clearance** After every step: 1 mm.



Danger of collision!

After each pass, the tool returns on a diagonal path before it advances for the next pass. If required, program an additional rapid traverse path to avoid a collision.

Undercut cycle G85

With the function G85, you can machine undercuts according to DIN 509 E, DIN 509 F and DIN 76 (thread undercut). The CNC PILOT determines the type of **undercut** using "K." For undercut parameters, see table.

The adjoining cylinder is machined if the tool is positioned at the cylinder diameter X "in front of" the cylinder.

The undercut roundings are executed with the radius $0.6 * I$.

Parameters

X, Z: Target point (X as diameter value)

I: Depth/wear allowance (radius)

■ DIN 509 E, F: Wear allowance – default: 0

■ DIN 76: Undercut depth

K: Undercut width and **type**

■ K no input: DIN 509 E

■ K=0: DIN 509 F

■ K>0: Undercut length for DIN 76

E: Reduced feed (for machining the undercut) – no input: Active feed rate



■ **Cutter radius compensation:** Not active.

■ **Allowances** are not considered.

Undercut according to DIN 509 E

Diameter	I	K	R
≤ 18	0.25	2	0.6
> 18 – 80	0.35	2.5	0.6
> 80	0.45	4	1

Undercut according to DIN 509 F

Diameter	I	K	R	P
≤ 18	0.25	2	0.6	0.1
> 18 – 80	0.35	2.5	0.6	0.2
> 80	0.45	4	1	0.3

Undercut angle for undercuts according to DIN 509 E and F: 15°

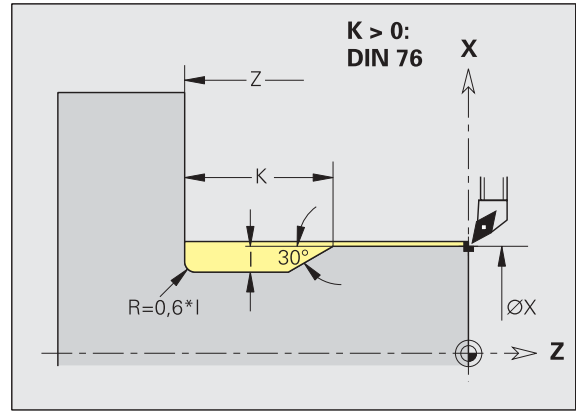
Transverse angle for an undercut according to DIN 509 F: 8°

I = depth of undercut

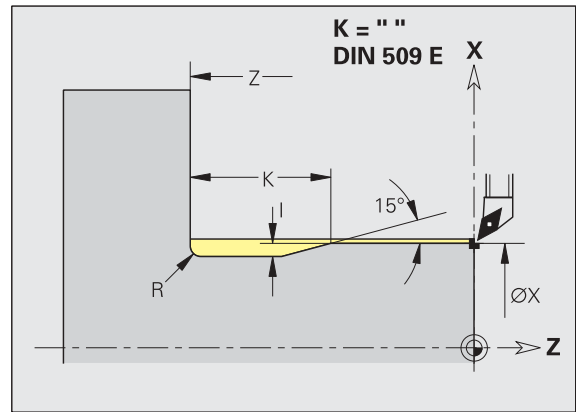
K = width of undercut

R = undercut radius

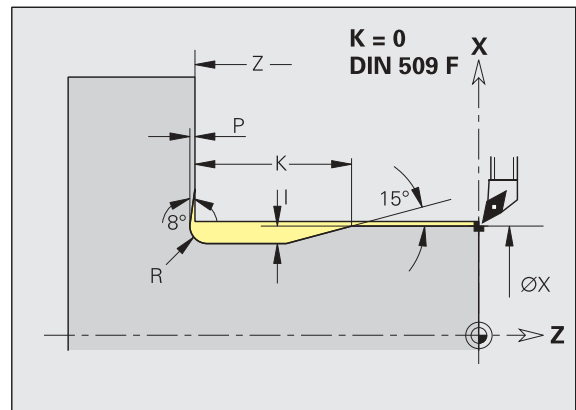
P = depth of end face



Undercut DIN 76 (thread undercut)



Undercut DIN 509 E



Undercut DIN 509 F

Recessing G86

G86 machines simple radial and axial recesses with chamfers. From the tool position the CNC PILOT calculates a radial/axial or an inside/outside recess.

"Oversize K" programmed: First pre-recessing, then finish recessing (finishing)

G86 machines chamfers at the sides of the recess. If you do not wish to cut the chamfers, you must position the tool at a sufficient distance from the workpiece. Calculate the starting position XS (diameter) as follows:

$$XS = XK + 2 * (1,3 - b)$$

XK: Contour diameter

b: Chamfer breadth

Cycle run

- 1 Calculate the cutting segmentation – maximum offset: SBF * cutting width (For SBF, see machining parameter 6).
- 2 Approach to clearance height at rapid traverse on paraxial path.
- 3 Execute the first cut - taking finishing allowance into account.
- 4 Without finishing allowance: "E" period of dwell.
- 5 Retract and approach for next pass.
- 6 Repeat 2 to 4 until the complete recess has been machined.
- 7 With finishing allowance: Finish-machine the recess.
- 8 Return to starting point at rapid traverse.

Parameters

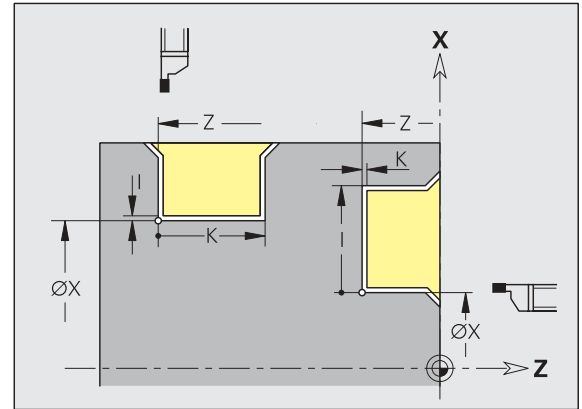
X, Z: Target point (X diameter)

Radial recess:

- I: Allowance
- I>0: Allowance (roughing and finishing)
 - I=0: No finishing
- K: Recess width – no input: a single cut is machined (recess width = tool width)

Axial recess:

- I: Recess width – no input: a single cut is machined (recess width = tool width)
- K: Allowance
- K>0: Allowance (roughing and finishing)
 - K=0: No finishing
- E: Period of dwell (for chip breaking) – default: Length of time for one revolution
- With finishing allowance: Only during finishing
 - Without finishing allowance: For each recess



- **Cutter radius compensation:** Active.
- **Allowances** are not calculated.

Cycle radius G87

G87 machines transition radii at orthogonal, paraxial inside and outside corners. The direction is taken from the "position/machining direction" of the tool.

A preceding longitudinal or transverse element is machined if the tool is located at the X or Z coordinate of the corner before the cycle is executed.

Parameters

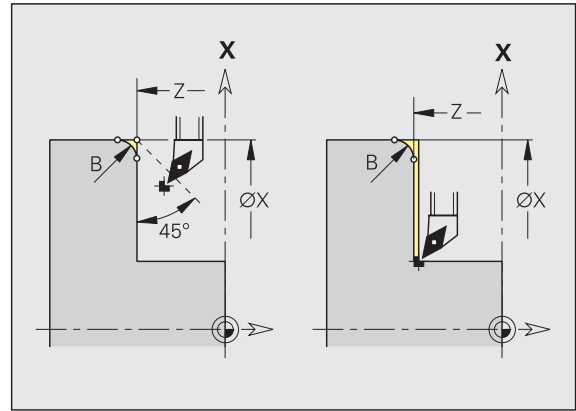
X, Z: Corner point (X diameter)

B Radius

E Reduced feed – default: Active feed



- **Cutter radius compensation:** Active.
- **Allowances** are not calculated.



Cycle chamfer G88

G88 machines chamfers at orthogonal, paraxial outside corners. The direction is taken from the "position/machining direction" of the tool.

A preceding longitudinal or transverse element is machined if the tool is located at the X or Z coordinate of the corner before the cycle is executed.

Parameters

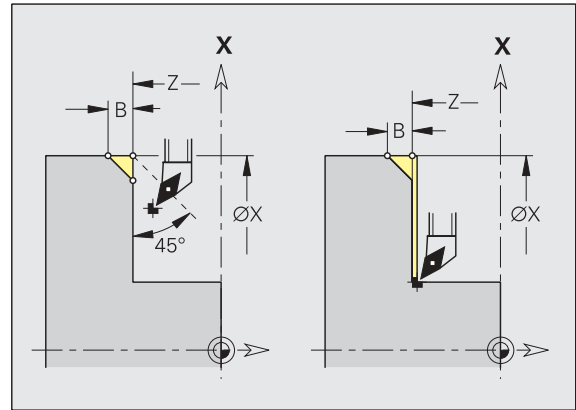
X, Z: Corner point (X diameter)

B Chamfer breadth

E Reduced feed – default: Active feed



- **Cutter radius compensation:** Active.
- **Oversizes** are not taken into account.



4.8 Thread Cycles

The slide requires a **run-in distance** to accelerate to the programmed feed rate before starting the actual thread and a **run-out distance** (overrun) at the end of the thread to decelerate again.

If the run-in/run-out length is too short, the thread may not attain the expected quality. In this case, the CNC PILOT displays a warning.

Starting length: $BA > 0.75 * (F*S)^2 / a + 0.15$

Thread cycle G31

G31 machine simple threads, successions of threads and multi-start threads with G24, G34 or G37 Geo.

External or internal threads are detected by the tool definition. The individual cuts are calculated from the thread depth, maximum approach "I" and type of approach "V."

Parameters

- NS: Block number (reference to basic element G1 Geo for successions of threads: block number of the first basic block)
- I: Maximum approach - maximum infeed distance
- B, P: Starting length, overrun length – no input: Length is calculated from adjacent undercuts or recesses. Undercut/recess does not exist: "thread starting length, thread runout length" from machining parameter 7.
- D: Cutting direction (reference: Defined direction for basic element) – default: 0;
 ■ D=0: Same direction
 ■ D=1: Opposite direction
- V: Type of feed – default: 0;
 ■ V=0: Constant cutting cross section for all cuts
 ■ V=1: Constant infeed
 ■ V=2: With distribution of remaining cut – first infeed = "remainder" of the thread depth/cutting depth division The last cut is divided into 1/2, 1/4, 1/8 and 1/8 cut.
 ■ V=3: Infeed is calculated from the pitch and the spindle speed
- H: Type of offset (infeed for smoothing the thread flank) – default: 0
 ■ H=0: No offset
 ■ H=1: Offset from left
 ■ H=2: Offset from right
 ■ H=3: Offset alternately right/left
- Q: Number of air cuts after the last cut (for reducing the cutting pressure in the thread base) – default: 0
- C: Starting angle (thread start is defined with respect to rotationally nonsymmetric contour elements) – default: 0



Danger of collision!

An excessive overrun length P might cause a collision. The overrun length can be checked during the simulation.

Runout length: $BE > 0.75 * (F*S)^2 / e + 0.15$

- BA: Minimum starting length
 BE: Minimum runout length
 F: Thread pitch in mm/revolution
 S: Spindle speed **in revolutions/seconds**
 a, e: Acceleration in mm/s²
 (see "Acceleration at block start/block end" in machine parameter 1105, ...)

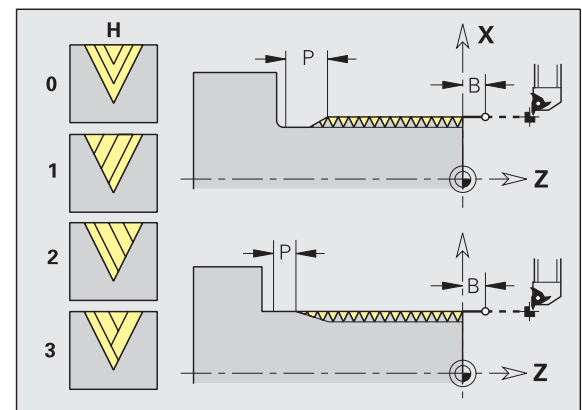
Cycle run

- 1 Calculate the cut segmentation.
- 2 Approach the "internal starting point" at rapid traverse on a diagonal path, which is calculated from "Slope length B" and the safety clearance.
- 3 Execute a thread cut.
- 4 Return at rapid traverse and approach for next pass.
- 5 Repeat 3 to 4 until the complete thread has been cut.
- 6 Execute idle cuts.
- 7 Return to "internal starting point."

For multiple threads, the same rate of cut is used for each thread turn, before the next infeed motion is executed.



- "Feed rate stop" becomes effective at the end of a thread cut.
- Feed rate override is not in effect.
- Do not use spindle override if the look-ahead is switched off!



Single thread G32

G32 cuts a simple thread in any desired direction and position (longitudinal, tapered or transverse thread; internal or external thread) without look-ahead function. G32 calculates the thread from the "thread end point," "thread depth" and the tool position. The main machining direction of the tool determines whether an internal or an external thread will be machined.

First infeed = "remainder" of the division of thread depth/cutting depth

Parameters

X, Z: End point of thread (X diameter)

F: Thread pitch

P: Thread depth

I: Maximum cutting depth

B: Remainder cuts – default: 0

■ B=0: Division of the "last cut" into 1/2, 1/4, 1/8 and 1/8 cut

■ B=1: No remaining cut division

Q: Number of air cuts after the last cut (for reducing the cutting pressure in the thread base) – default: 0

K: Run-out length at thread end – default: 0

W: Taper angle (range: $-45^\circ < W < 45^\circ$) – default: 0; position of the taper thread with reference to longitudinal or transverse axis.

■ W>0: Rising contour (in machining direction)

■ W<0: Falling contour

C: Starting angle (thread start is defined with respect to rotationally nonsymmetric contour elements) – default: 0

H: Type of tool offset (offset of the individual approaches to smooth the thread flanks) – default: 0

■ H=0: No offset

■ H=1: Offset to the left

■ H=2: Offset to the right

■ H=3: Offset alternating left and right

Cycle run

- 1 Calculate the cut segmentation.
- 2 Execute a thread cut.
- 3 Return at rapid traverse and approach for next pass.
- 4 Repeat 3 to 4 until the complete thread has been cut.
- 5 Execute idle cuts.
- 6 Return to starting point.



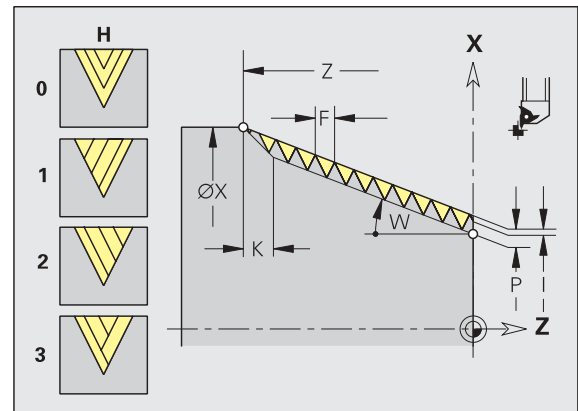
■ "Feed rate stop" becomes effective at the end of a thread cut.

■ Feed rate override is not in effect.

■ Spindle override is not in effect.

■ Make thread with G95 (feed rate per revolution).

■ **Look-ahead** is switched off.



Thread single path G33

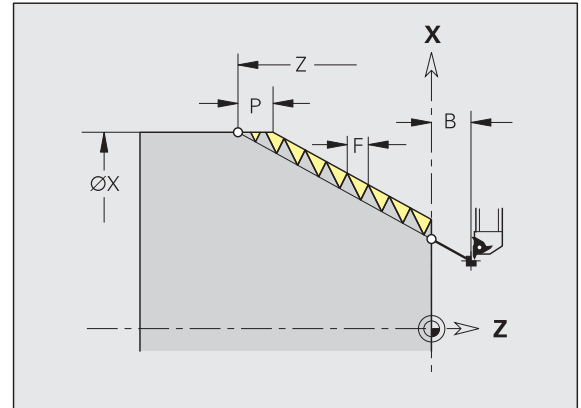
G33 cuts threads in any desired direction and position (longitudinal, tapered or transverse threads; internal or external threads). G33 uses a single pass to cut a thread starting at the tool position and ending at "X, Z." (Spindle and feed drives are synchronized during thread cutting.)

Parameters

- X, Z: Diameter, length to end point of thread (X diameter)
- F: Feed per revolution (thread pitch)
- B, P: Run-in length, overrun length – default: 0 (see "4.8 Thread Cycles")
- C: Starting angle (thread start is defined with respect to rotationally nonsymmetric contour elements) – default: 0
- Q: Number of the spindle
- H: Reference direction for spindle pitch - default: 0
 - H=0: Feed rate in Z axis (for longitudinal and tapered threads up to a maximum angle of $+45^\circ/-45^\circ$ to Z axis)
 - H=1: Feed rate in X axis (for transverse and taper threads up to a maximum angle of $+45^\circ/-45^\circ$ to the X- axis)
 - H=2: Feed rate in Y axis
 - H=3: Contouring feed rate
- E: Variable pitch – default: 0
 - E=0: Constant pitch
 - E>0: Increase pitch per revolution by E
 - E<0: Decrease pitch per revolution by E



- "Feed rate stop" becomes effective only at the end of a thread cut.
- Feed rate override is not in effect.
- Do not use spindle override if the look-ahead is switched off!
- Make thread with G95 (feed rate per revolution).



4.9 Drilling cycles

Simple drilling cycle G71

G71 is used for axial and radial bore holes using driven or stationary tools.

The cycle is used for:

- Individual bore hole without contour description
- Bore holes with contour definition (individual bore hole or hole pattern) in the following program sections:
 - FRONT
 - REAR SIDE
 - SURFACE

The type of drill determines when a feed rate reduction begins:

- Indexable inserts and twist drills with 180° drilling angle:
Reduction at end of hole – 2 * safety clearance
- Other drills:
Drill end – length of first cut – safety clearance
(length of first cut=drill tip; safety clearance: see machining parameter 9 "Drilling" or G47, G147).

Parameters

- NS: Contour block number with geometry of bore hole (G49, G300/G310-Geo) – no input: Individual bore hole without contour definition
- X, Z: Position, length – end point of axis/radial holes (X diameter value)
- E: Period of dwell in seconds (for chip breaking at end of hole) – default: 0
- V: Feed rate reduction (50%) – default: 0
 - V=0 or 2: Reduction at start
 - V=1 or 3: Reduction at start and at end
 - V=4: Reduction at end
 - V=5: No reduction
- Exception with V=0 and V=1:** No reduction when boring when indexable insert drills and twist drills with 180° drilling angle are used
- D: Retraction speed – default: 0
 - D=0: Rapid traverse
 - D=1: Feed rate
- K: Retraction plane (radial holes, holes in the YZ plane: diameter) – default: to starting position or to safety clearance

Cycle run

1 Bore hole without contour definition

Precondition: Tool is located at the safety distance from the bore hole ("starting position").

Bore hole with contour definition:

The tool approaches the starting position at rapid traverse:

- If K is not programmed: Approach to clearance height
- If K is programmed: Approach to K, and then to clearance height.

2 Start drilling bore hole – feed rate reduction according to "V"

3 Drill hole at feed rate.

4 Drill through – feed rate reduction according to "V"

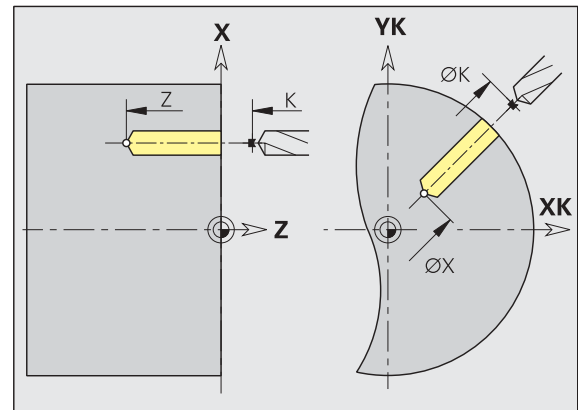
5 Retract at rapid traverse or feed rate according to "D"

6 Position to which the tool retracts:

- If K is not programmed: Retraction to the starting point
- If K is programmed: Retraction to position K



- Single hole without contour description: Program „X or Z“ as alternative.
- Hole with contour description: Do not program X, Z.
- Hole pattern: NS refers to the bore hole contour (and not the definition of the pattern)).



Boring, countersinking G72

Use of G72: Boring, sinking, reaming, NC spot drilling or centering for axial/ radial holes with stationary or driven tools.

G72 is used for bore holes with contour definition (individual bore hole or hole pattern) in the following program sections:

- FRONT
- REAR SIDE
- SURFACE

Parameters

NS: Contour block number with geometry of bore hole (G49, G300/ G310-Geo)

E: Period of dwell (for chip breaking at end of hole) – default: 0

D: Retraction speed – default: 0

- D=0: Rapid traverse
- D=1: Feed rate

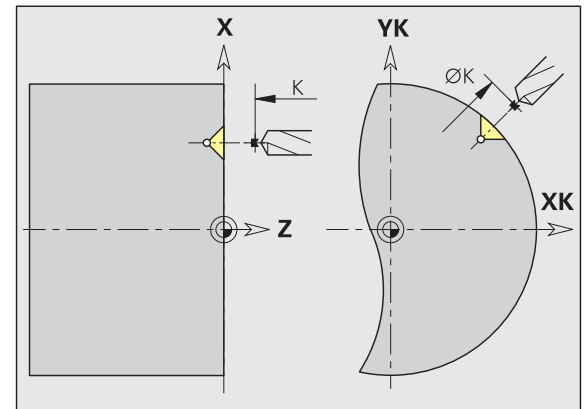
K: Retraction plane (radial holes, holes in the YZ plane: diameter) – default: to starting position or to safety clearance

Cycle run

- 1 Approach starting position at rapid traverse according to "K":
 - K not programmed: Approach to clearance height.
 - K programmed: Approach to "K," and then to clearance height.
- 2 Start drilling bore hole at reduced feed rate (50%).
- 3 Move at feed rate to end of bore hole.
- 4 Retract at rapid traverse or feed rate according to "D."
- 5 Position to which tool retracts depends on "K":
 - K not programmed: Retract to starting position.
 - K programmed: Retract to position "K."



Hole pattern: "NS" refers to the bore hole contour (and not the definition of the pattern).



Tapping G73

G73 cuts axial/radial threads using driven or stationary tools.

G73 is used for bore holes with contour definition (individual bore hole or hole pattern) in the following program sections:

- FRONT
- REAR SIDE
- SURFACE

The starting position is calculated from the safety distance and the "run-in (slope) length B."

Meaning of "retraction length J": Use this parameter for floating tap holders. The cycle calculates a new nominal pitch on the basis of the thread depth, the programmed pitch, and the "retract length." The nominal pitch is somewhat smaller than the pitch of the tap. During tapping, the drill is pulled away from the chuck by the "retraction length." With this method you can achieve higher service life from the taps.

Parameters

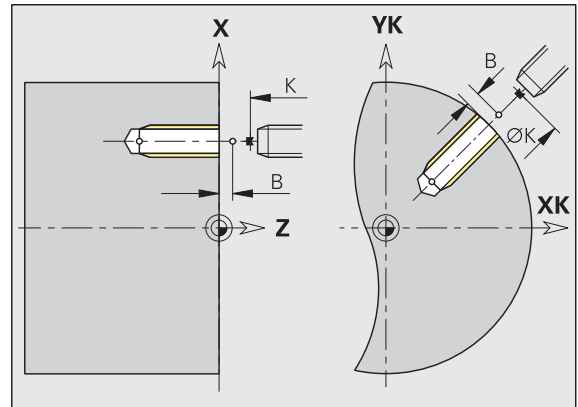
- NS: Contour block number with geometry of bore hole (G49, G300/G310-Geo)
- B: Slope length – default: Machining Parameter 7 "Thread starting length [GAL]"
- S: Retraction speed – default: Tapping speed
- K: Retraction plane (radial holes, holes in the YZ plane: diameter) – default: to starting position or to safety clearance
- J: Retraction length when using floating tap holders – default: 0

Cycle run

- 1 Approach starting position at rapid traverse according to "K":
 - K not programmed: Approach directly to starting position.
 - K programmed: Approach to "K," and then to starting position.
- 2 Move along "slope length B" at feed rate (synchronization of spindle and feed drives).
- 3 Cut the thread.
- 4 Returns at "return speed S" according to "K":
 - K not programmed: Return to starting point.
 - K programmed: Return to position "K."



- Hole pattern: NS refers to the bore hole contour (and not the definition of the pattern)).
- "Cycle stop" becomes effective at the end of a thread cut.
- Feed rate override is not in effect.
- Do not use spindle override!



Thread cycle G36

G36 cuts axial/radial threads using driven or stationary tools. Depending on X/Z, G36 decides whether a radial or axial thread will be machined.

Move to the starting point before G36. G36 returns to the starting position after having cut the thread.

Parameters

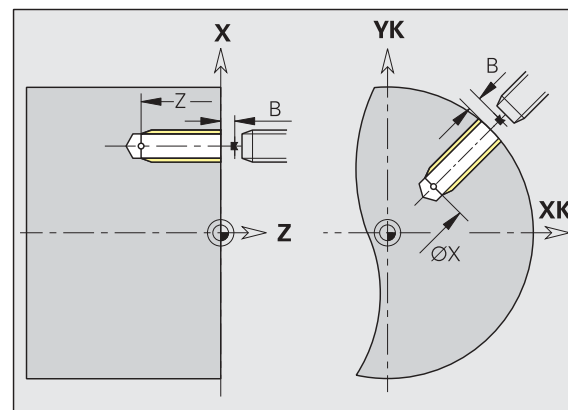
- X: Diameter – end point for axial holes
- Z: Length – end point for radial holes
- F: Feed per revolution - thread pitch
- Q: Number of spindle - default: 0 (main spindle)
- B: Run-in length for synchronizing spindle and feed drive (see G33)
- H: Reference direction for thread pitch – default: 0
 - H=0: Feed rate on the Z axis
 - H=1: Feed rate on the X axis
 - H=2: Feed rate on the Y axis
 - H=3: Contour feed rate
- S: Retraction speed (higher speed for the retraction) – default: Same speed as for tapping

Types of taps:

- Stationary tap: Spindle and feed drives are synchronized.
- Driven tap: Driven tool (auxiliary spindle) and feed drive are synchronized.



- "Cycle stop" becomes effective at the end of a thread cut.
- Feed rate override is not in effect.
- Do not use spindle override!
- Use a floating tap holder if the driven tool is not controlled, e.g. by a ROD encoder.



Deep-hole drilling G74

G74 is used for axial and radial bore holes in several stages using driven or stationary tools.

"1st drilling depth P" is used for the first pass. The CNC PILOT then automatically reduces the drilling depth with each subsequent pass by the "reducing value I", however, without falling below the "minimum hole depth J." After each pass, the tool is retracted either by "return distance B" or to the starting point of the hole.

The cycle is used for:

- Individual bore hole without contour description
- Bore holes with contour definition (individual bore hole or hole pattern) in the following program sections:
 - FRONT
 - REAR SIDE
 - SURFACE

The type of drill determines when a feed rate reduction begins:

- Indexable inserts and twist drills with 180° drilling angle:
Reduction at end of hole – 2 * safety clearance
- Other drills:
Drill end – length of first cut – safety clearance
(length of first cut=drill tip; safety clearance: see machining parameter 9 "Drilling" or G47, G147).

Parameters

- NS: Contour block number with geometry of bore hole (G49, G300/ G310-Geo) – no input: Individual bore hole without contour definition
- X, Z: Position, length – end point of axis/radial holes (X diameter value)
- P: 1st drilling depth
- I: Reduction value – default: 0
- B: Retraction distance – default: to starting point of hole
- J: Minimum hole depth - default: 1/10 of P
- E: Period of dwell (for chip breaking at end of hole) – default: 0
- V: Feed rate reduction (50%) – default: 0
- V=0 or 2: Reduction at start
 - V=1 or 3: Reduction at start and at end
 - V=4: Reduction at end
 - V=5: No reduction
- Exception with V=0 and V=1:** No reduction when boring when indexable insert drills and twist drills with 180° drilling angle are used
- D: Retraction speed and infeed within the hole – default: 0
- D=0: Rapid traverse
 - D=1: Feed rate
- K: Retraction plane (radial holes, diameter) – default: to starting position or to safety clearance

Cycle run

1 Bore hole without contour definition

Precondition: Tool is located at the safety distance from the bore hole ("starting position").

Bore hole with contour definition

The tool approaches the starting position at rapid traverse according to "K":

- K not programmed: Approach to clearance height.
- K programmed: Approach to "K," and then to clearance height.

2 Start drilling bore hole – feed rate reduction according to "V."

3 Drill hole in several passes.

4 Drill through – feed rate reduction according to "V."

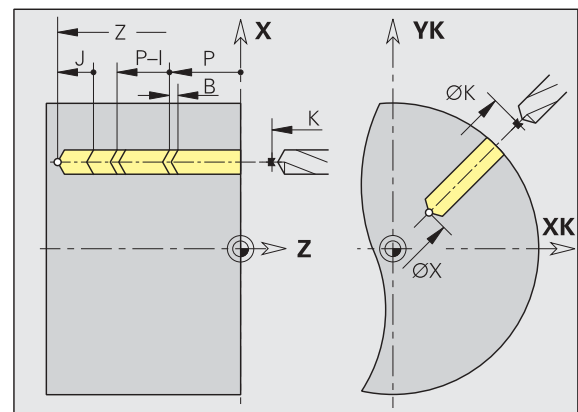
5 Retract at rapid traverse or feed rate according to "D."

6 Position to which tool retracts depends on "K":

- K not programmed: Retract to starting position.
- K programmed: Retract to position "K."



- Single hole without contour description:
Program +X or Z+ as alternative.
- Hole with contour description: Do not program X, Z.
- Hole pattern: NS refers to the bore hole contour (and not the definition of the pattern)).
- A "feed rate reduction at end" goes into effect only at the last drilling stage.



4.10 C-Axis Machining

4.10.1 General C-Axis Functions

Select C axis G119

If several C axes are available, use G119 to select a C axis and to switch the active C axis during machining.

G119 assigns the C axis entered in "Q" to the slide. Before assigning an active C axis to another slide, cancel the previous assignment with G119 without Q.

Parameters

Q: Number of the C axis – default: 0

■ Q=0: Cancel the current assignment of C axis to slide.

■ Q>0: Assign the C axis to a slide.

Reference diameter G120

G120 determines the reference diameter of the unrolled lateral surface. Program G120 if you use "CY" for G110 to G113. G120 is a modular function.

Parameters

X: Diameter

Zero point displacement, C axis G152

G152 defines a zero offset for the C axis (reference: machine parameter 1005, ff "reference point C axis). The zero point is valid until the end of the program.

Parameters

C: Angle of the "new" C-axis zero point

Standardize C axis G153

G153 resets a traverse angle $>360^\circ$ or $<0^\circ$ to the corresponding angle modulo 360° – without moving the C axis.



G153 is only used for lateral-surface machining. An automatic modulo 360° function is carried out on the end faces.

4.10.2 Front/Rear Face Machining

Rapid traverse on end face G100

The tool moves at rapid traverse along the shortest path to the end point.

Parameters

- X: Diameter of the end point
 C: Final angle (angular dimension)
 XK, YK: End point in Cartesian coordinates
 Z: End point - default: Current Z position



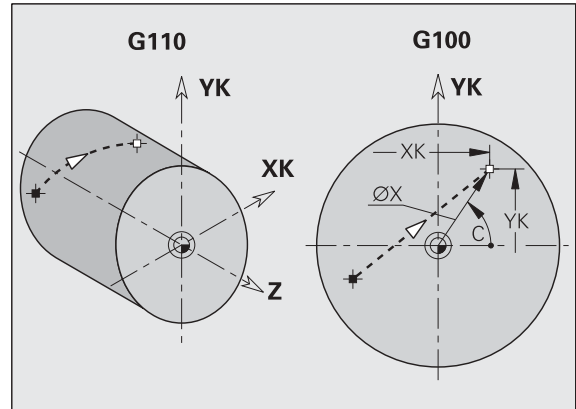
Programming

- **X, C, XK, YK, Z:** Absolute, incremental or modal.
- Program either X and C or XK and YK.



Danger of collision!

During G100 the tool moves on a linear path. To position the workpiece to a defined angle, use G110.



Linear segment on end face G101

The tool moves linearly at the feed rate to the "end point."

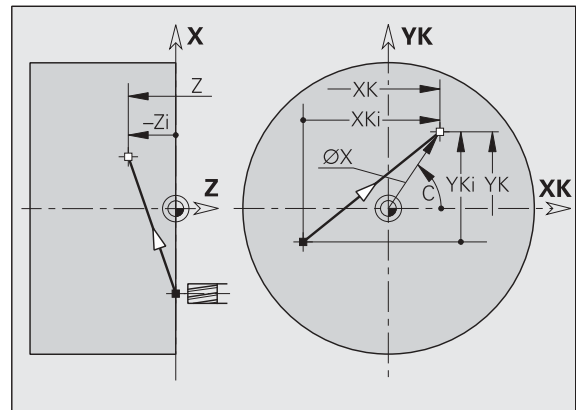
Parameters

- X: Diameter of the end point
 C: Final angle (angular dimension)
 XK, YK: End point in Cartesian coordinates
 Z: End depth – default: Current Z position



Programming

- **X, C, XK, YK, Z:** Absolute, incremental or modal.
- Program either X and C or XK and YK.



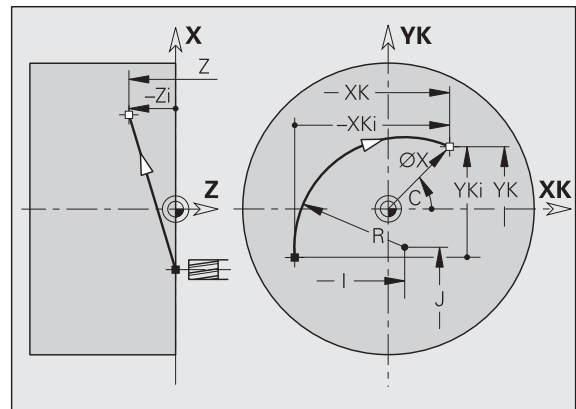
Circular arc on front face/rear face G102/G103

The tool moves in a circular arc at the feed rate to the "end point."

Direction of rotation: see help graphic

If you program "H=2 or H=3", you can machine linear slots with a circular base. If

- H=2: With I and K
- H=3: Define the circle center with J and K.



Circular arc G102

Continued ►

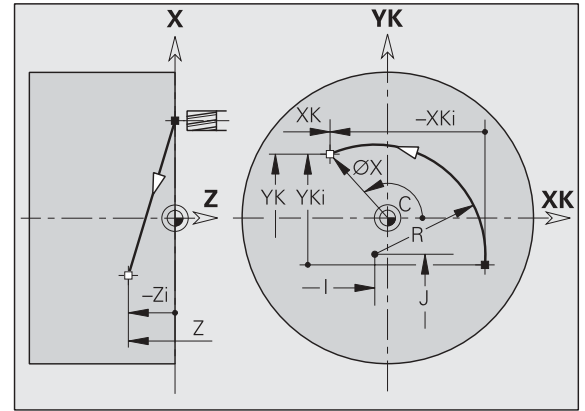
Parameters

- X: Diameter of the end point
 C: Final angle (angular dimension)
 XK, YK: End point in Cartesian coordinates
 R: Radius
 I, J: Center in Cartesian coordinates
 Z: End depth – default: Current Z position
 H: Circular plane (machining plane) – default: 0
 ■ H=0, 1: Facing (XY plane)
 ■ H=2: Machining in YZ plane
 ■ H=3: Machining in XZ plane
 K: Center point (Z direction) – only for H=2, 3



Programming

- **X, C, XK, YK, Z:** absolute, incremental or modal
- **I, J:** absolute or incremental
- Program either X–C or XK–YK
- Program either “center” or “radius”
- With “radius”: circular arcs possible only $\leq 180^\circ$
- End point in the coordinate origin: Program XK=0 and YK=0



Circular arc G103

4.10.3 Lateral Surface Machining

Rapid traverse on lateral surface G110

The tool moves at rapid traverse along the shortest path to the end point.

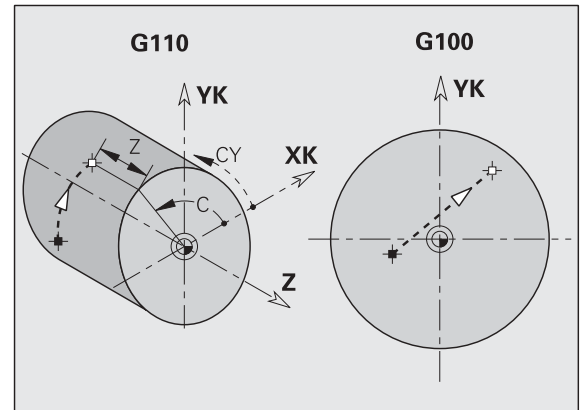
Parameters

- Z: End point
 C: Final angle (angular dimension)
 CY: End point as linear value (referenced to unrolled reference diameter G120)
 X: End point (diameter)



Programming

- **Z, C, CY:** absolute, incremental or modal
 - Program either Z–C or Z–CY
- Use G110** to position the C axis to a defined angle (programming: N.. G110 C...).



Linear segment on lateral surface G111

The tool moves linearly at the feed rate to the "end point."

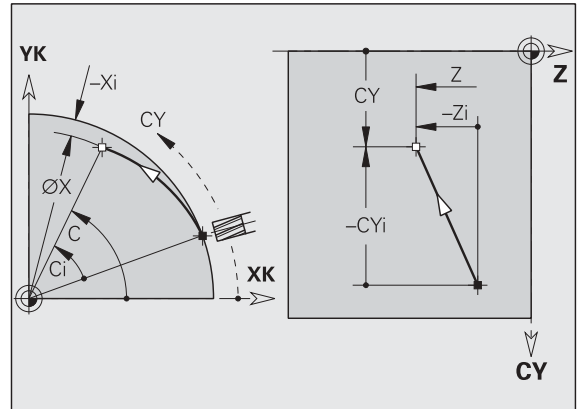
Parameters

- Z: End point
C: Final angle (angular dimension)
CY: End point as linear value (referenced to unrolled reference diameter G120)
X: End depth (diameter value) – default: Current X position



Programming

- **Z, C, CY:** Absolute, incremental or modal.
- Program either Z and C, or Z and CY.



Circular arc on lateral surface G112/G113

The tool moves in a circular arc at the feed rate to the "end point."

Direction of rotation: see help graphic

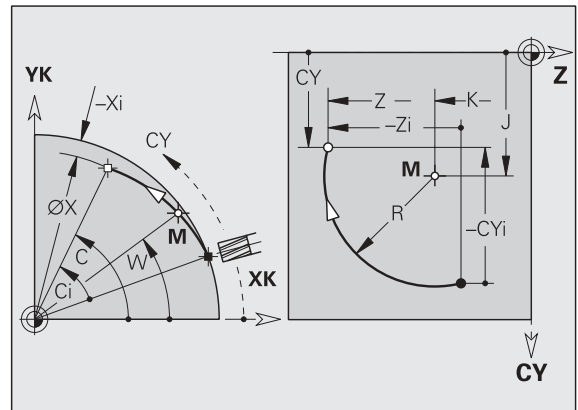
Parameters

- Z: End point
C: Final angle (angular dimension)
CY: End point as linear value (referenced to unrolled reference diameter G120)
R: Radius
K, W: Position, angle to midpoint
J: Center point coordinate as a linear value (referenced to unrolled G120 reference diameter)
X: End depth (diameter value) – default: Current X position

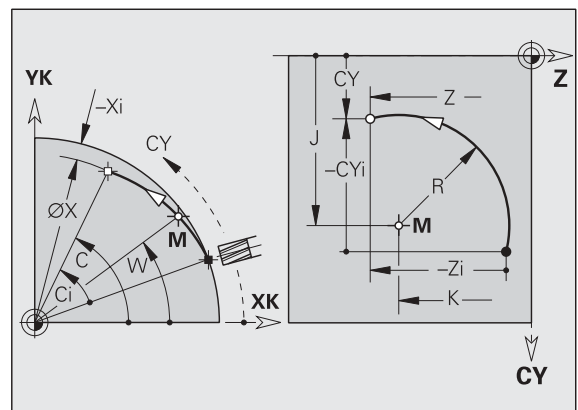


Programming

- **Z, C, CY:** absolute, incremental or modal
- **K, W, J:** Absolute or incremental
- Program either Z-C and K-W or Z-CY and K-J
- Program either "center" or "radius"
- With "Radius": Only circular arcs <= 180° are possible.



Circular arc G112



Circular arc G113

4.11 Milling Cycle Group

Contour milling G840

G840 mills, finishes, engraves or deburrs figures or "free contours" (open or closed) in the following program sections:

- FRONT
- REAR SIDE
- SURFACE

NS/NE defines the contour section and the contour direction. NE is not programmed for closed contours. For a single contour element, you can program NS **and** NE to reverse the contour direction.

You can change the **machining direction** and the **cutter radius compensation (TRC)** with the "cycle type Q," the "cutting direction H" and the rotational direction of the tool (see following table).

Deburring

G840 deburrs if "chamfer width B" is programmed. The "milling depth P" defines the plunging depth of the tool – preempting the "infeed I."

"Preparation diameter J" (see illustration):

- Open contour – J programmed: The contour is deburred all around. Prerequisite: The deburring tool has a smaller diameter than the milling cutter.
- Open contour – deburring tool and milling cutter have equal diameter: J is omitted
- Closed contour: The side programmed with "cycle type Q" is deburred; J is omitted.

Other parameters are usually programmed in the same way as milling contours.

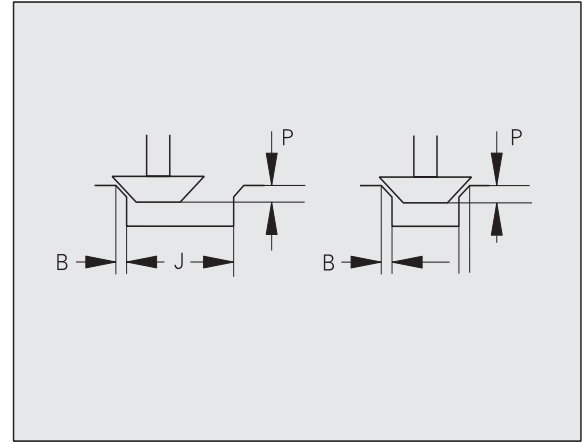
Approach and departure

For closed contours, the point of the surface normal from the tool position to the first contour element is the point of approach and departure. If no surface normal intersects the tool position, the starting point of the first element is the point of approach and departure.

For figures, you can select the approach/depart element or machine parts of the figure by selecting "Begin, End Element Number D, V."

Oversize

A G58 oversize "shifts" the contour to be milled in the direction given in "cycle type." "Inside milling" (closed contour) contracts the contour, while an "outside milling" type expands it. For open contours, the contour is shifted to the left or right, depending on the cycle type.



Cycle run

- 1 Starting position (X, Z, C) is the position before the cycle begins.
- 2 Calculate the milling depth infeeds.
- 3 Move to the safety clearance and plunge to the first milling depth.
- 4 Mill the contour.
- 5 ■ For open contours and for slots whose width = tool diameter: Plunge to the next milling depth and mill the contour in the opposite direction.
■ For closed contours and slots: Retract by the safety clearance, return and cut to the next milling depth.
- 6 Repeat steps 4 and 5 until the complete contour is milled.
- 7 Retract to "return plane K."

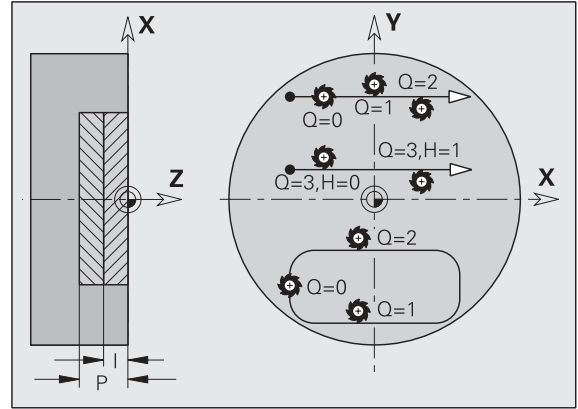


- With "cycle type Q=0," oversizes are not taken into account.
- G57 and negative G58 oversizes are not taken into account.

Continued ►

Parameters

- Q: Cycle type (= milling location)
- Q=0: Milling center on the contour (without milling cutter radius compensation)
 - Q=1—closed contour: inside milling
 - Q=1—with open contour: left in machining direction; intersecting areas which are programmed in directly successive contour elements are **not** machined.
 - Q=2—with closed contour: outside milling
 - Q=2—with open contour: right in machining direction; Intersecting areas which are programmed in directly successive contour elements are **not** machined.
 - Q=3 (only with open contours): The tool cuts from the left or right of the contour depending on the “cutting direction H” **and** the direction of tool rotation (see following table).
 - Q=4—with closed contour: inside milling
 - Q=4—with open contour: left in machining direction;; Intersecting areas which are programmed in directly successive contour elements are machined
 - Q=5—with closed contours: outside milling
 - Q=5—with open contours: right in machining direction; Intersecting areas which are programmed in directly successive contour elements are machined.
- NS: Block number—beginning of contour section
- Figures: Block number of the figure.
 - “Free contour”: First contour element (not starting point).
- NE: Block number—end of contour section
- Figures, closed contours: No input
 - Open contour: Last contour element
 - Contour consists of one element: Input unnecessary
- H: Cutting direction—default: 0
- H=0: Up-cut milling
 - H=1: Climb milling
- I: (Maximum) infeed—default: Milling in one infeed
- F: Infeed rate (depth infeed)—default: Active feed rate
- E: Reduced feed rate for circular elements—default: Current feed rate
- R: Radius of approaching/departing arc—default: 0
- R=0: Contour element is approached directly; feed to starting point above the milling plane—then vertical plunge.
 - R>0: Tool moves on an approaching/departing arc that connects tangentially to the contour element.
 - R<0 for inside corners: Tool moves on an approaching/departing arc that connects tangentially to the contour element.
 - R<0 for outside corners: Contour element is approached/ departed on a tangentially linear path
- P: Milling depth
- Milling, finishing—default: Milling depth from the contour description
 - Deburring: Plunging depth of the tool.

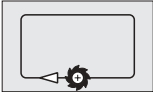
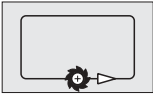
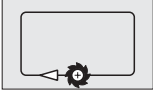
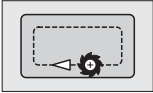
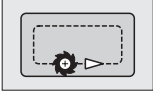
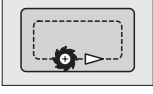
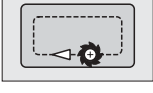
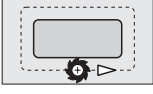
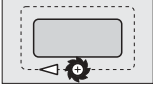
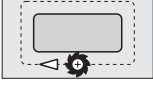


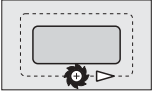


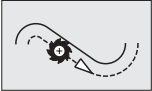

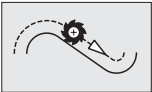
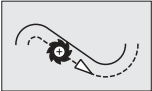
- K: Retraction plane—default: return to starting position
- Front or rear face: Retraction position in Z direction
 - Lateral surface: Retraction position in X direction (diameter)
- B: Chamfer width for deburring the upper edges (sign has no effect).
- J: Preparation diameter (tool diameter from machining):
- Required for deburring of open contours.
 - Not required, if diameter of deburring tool = diameter of milling tool.
- D, V: Beginning, end of element number for figures (only if partial figures are machined).

Element numbers for figures:

Direction of contour definition for figures:
Counterclockwise.

- **Rectangles, polygons and linear slots:** The “angle of orientation” (angle with respect to the longitudinal axis, or to one side of a polygon) points to the first contour element.
- **Circular slot:** The larger arc is the first contour element.
- **Full circle:** The upper semicircle is the first contour element.

Closed contours				
Cycle type	Cutting direction	Direction of tool rotation	TRC	Version
Contour (Q=0)	–	Mx03	–	
Contour	–	Mx03	–	
Contour	–	Mx04	–	
Contour	–	Mx04	–	
Inside (Q=1)	Up-cut milling (H=0)	Mx03	Right	
Inside	Up-cut milling (H=0)	Mx04	Left	
Inside	Climb milling (H=1)	Mx03	Left	
Inside	Climb milling (H=1)	Mx04	Right	
Outside (Q=2)	Up-cut milling (H=0)	Mx03	Right	
Outside	Up-cut milling (H=0)	Mx04	Left	
Outside	Climb milling (H=1)	Mx03	Left	

Closed contours				
Cycle type	Cutting direction	Direction of tool rotation	TRC	Version
Outside	Climb milling (H=1)	Mx04	Right	
Contour (Q=0)	–	Mx03	–	
Contour	–	Mx04	–	
Right (Q=3)	Up-cut milling (H=0)	Mx03	Right	
Left (Q=3)	Up-cut milling (H=0)	Mx04	Left	
Left (Q=3)	Climb milling (H=1)	Mx03	Left	
Right (Q=3)	Climb milling (H=1)	Mx04	Right	

Pocket milling – roughing G845

G845 roughs closed contours and figures in the following program sections:

- FRONT
- REAR SIDE
- SURFACE

You can change the **cutting direction** with the "cutting direction H," the "machining direction Q" and the direction of tool rotation (see table G846).

Parameters

- NS: Block number – reference to contour description
P: (Maximum) milling depth (infeed in the working plane)
I: Oversize in X direction
K: Oversize in Z direction
U: (Minimum) overlap factor – overlap of tool paths (overlap = $U \cdot$ cutter diameter) – default: 0.5
V: Overrun factor – no significance for machining with the C axis
H: Cutting direction – default: 0
■ H=0: Up-cut milling
■ H=1: Climb milling
F: Feed rate for infeed – default: Active feed rate
E: Reduced feed rate for circular elements – default: Current feed rate
J: Retraction plane – default: return to starting position
■ Front or rear face: Retraction position in Z direction
■ Lateral surface: Retraction position in X direction (diameter)
Q: Machining direction – default: 0
■ Q=0: From the inside toward the outside
■ Q=1: From the outside toward the inside

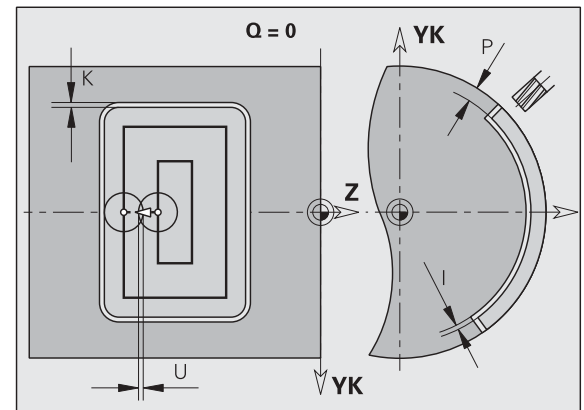
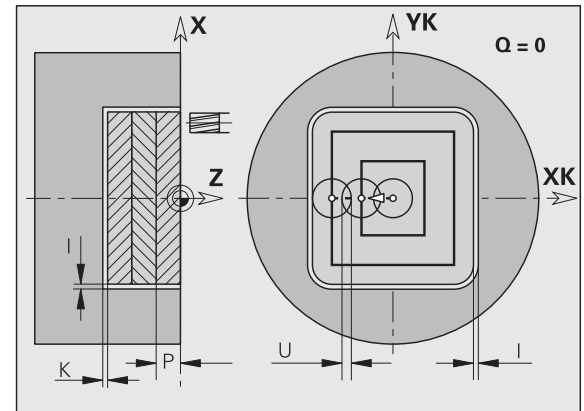
Programming in the Y axis See "CNC PILOT 4290 with Y Axis" User's Manual.

Cycle run

- 1 Starting position (X, Z, C) is the position before the cycle begins.
- 2 Calculate the cutting segmentation (infeeds to the working planes, infeeds in the working plane).
- 3 Move to the safety clearance and plunge to the first milling depth.
- 4 Mill the first plane.
- 5 Retract by the safety clearance, return and cut to the next milling depth.
- 6 Repeat steps 4 and 5 until the complete surface is milled.
- 7 Retract to "return plane J."



Oversizes are taken into account with G845 (G57: X, Z direction; G58: equidistant oversize in the milling plane).



Pocket milling, finishing G846

G846 finishes closed contours and figures in the following program sections:

- FRONT
- REAR SIDE
- SURFACE

You can change the **cutting direction** with the "cutting direction H," the "machining direction Q" and the direction of tool rotation (see following table).

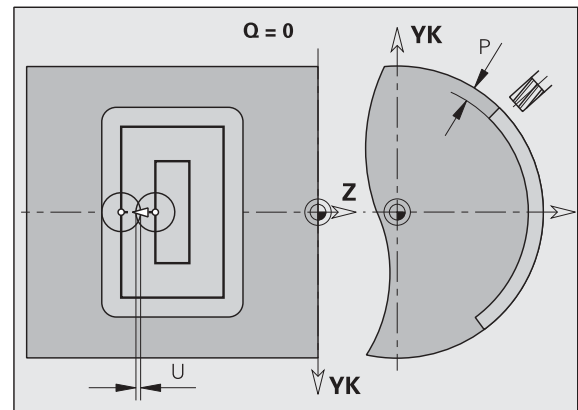
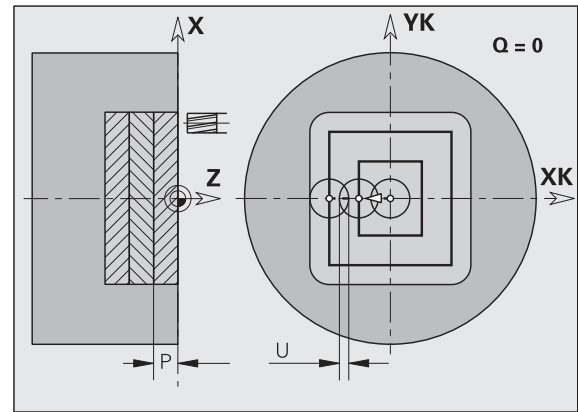
Parameters

- NS: Block number – reference to contour description
- P: (Maximum) milling depth (infeed in the working plane)
- R: Radius of approaching/departing arc – default: 0
- R=0: Contour element is approached directly; feed to approach point above the milling plane – then perpendicular feed to plunging depth.
 - R>0: Mill moves along approaching/departing arc, which connects tangentially with the contour element.
- U: (Minimum) overlap factor – overlap of tool paths (overlap = U * cutter diameter) – default: 0.5
- V: Overrun factor – no significance for machining with the C axis
- H: Cutting direction – default: 0
- H=0: Up-cut milling
 - H=1: Climb milling
- F: Feed rate for infeed – default: Active feed rate
- E: Reduced feed rate for circular elements – default: Current feed rate
- J: Retraction plane – default: return to starting position
- Front or rear face: Retraction position in Z direction
 - Lateral surface: Retraction position in X direction (diameter)
- Q: Machining direction – default: 0
- Q=0: From the inside toward the outside
 - Q=1: From the outside toward the inside

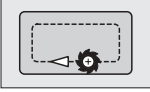
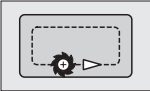
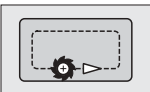
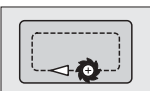
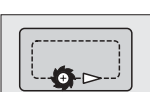
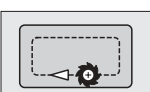
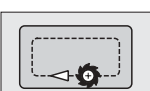
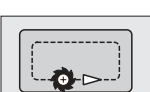
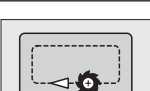
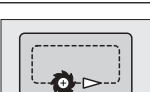

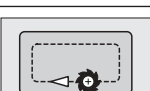
Programming in the Y axis See "CNC PILOT 4290 with Y Axis" User's Manual.

Cycle run

- 1 Starting position (X, Z, C) is the position before the cycle begins.
- 2 Calculate the cutting segmentation (infeeds to the working planes, infeeds in the working plane).
- 3 Move to the safety clearance and plunge to the first milling depth.
- 4 Mill the first plane.
- 5 Retract by the safety clearance, return and cut to the next milling depth.
- 6 Repeat steps 4 and 5 until the complete surface is milled.
- 7 Retract to "return plane J."



Continued ►

Pocket milling				
Cycle	Cutting direction	Machining direction	Direction of tool rotation	Version
G845 G846	Up-cut milling (H=0) Up-cut milling (H=0)	from inside (Q=0) –	Mx03 Mx03	
G845 G846	Up-cut milling (H=0) Up-cut milling (H=0)	from inside (Q=0) –	Mx04 Mx04	
G845	Up-cut milling (H=0)	from outside (Q=1)	Mx03	
G845	Up-cut milling (H=0)	from outside (Q=1)	Mx04	
G845 G846	Climb milling (H=1) Climb milling (H=1)	from inside (Q=0) –	Mx03 Mx03	
G845 G846	Climb milling (H=1) Climb milling (H=1)	from inside (Q=0) –	Mx04 Mx04	
G845	Climb milling (H=1)	from outside (Q=1)	Mx03	
G845	Climb milling (H=1)	from outside (Q=1)	Mx04	
G846	Up-cut milling (H=0)	–	Mx03	
G846	Up-cut milling (H=0)	–	Mx04	
G846	Climb milling (H=1)	–	Mx03	
G846	Climb milling (H=1)	–	Mx04	

4.12 Special functions

4.12.1 Chucking Equipment in Simulation

Clamping G65

G65 displays the selected chucking equipment in the simulation graphics. G65 needs to be programmed separately for each chuck. G65 H.. without X, Z cancels the chuck in the simulation graphics.

Chucks are described in the database and are defined in CHUCKING EQUIPMENT (H=1..3).

Parameters

- H: Chuck number (H=1..3: Reference to CHUCKING EQUIPMENT)
- X, Z: Starting point – position of the chuck reference point (X diameter) – **Reference: Workpiece zero point**
- D: Spindle number (reference: CHUCKING EQUIPMENT section)
- Q: Chuck shape (only for chuck jaws) – default: Q from the CHUCKING EQUIPMENT section.

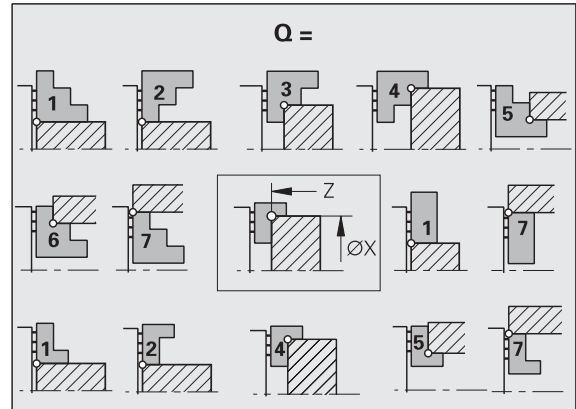
Chuck reference point

"X, Z" defines the position of the chucking equipment in the simulation graphics. The position of the reference point depends on the clamping form (see illustration).

The CNC PILOT mirrors the chucks "H=1 to 3" if they are positioned to the right of the workpiece.

Note on the simulation graphics and the reference point:

- H=1 – chuck
 - "Open" chuck is displayed.
 - Reference point X: Chuck center.
 - Reference point Z: Right edge (account for width of chuck jaws).
- H=2 – chuck jaw ("chuck shape Q" defines the reference point and internal/external clamping)
 - Position of the reference point: see illustration at upper right
 - Internal clamping: 1, 5, 6, 7
 - External clamping: 2, 3, 4
- H=3 – chucking accessory (dead center, lathe center, etc.)
 - Reference point in X: Chuck center
 - Reference point in Z: Chuck tip



Program NC blocks containing G65 with "slide code \$.." if your lathe has more than one slide. Otherwise, the system displays more than one chuck.

Example: Chucking equipment

```

...
SPANNMITTEL 1 [CHUCKING EQUIPMENT]
H1 ID"KH110" [Chuck]
H2 ID"KBA250-77" [Chuck jaw]
H4 ID"KSP-601N" [Lathe center]
...
ROHTEIL [BLANK]
N1 G20 X80 Z200 K0
...
BEARBEITUNG [MACHINING]
$1 N2 G65 H1 X0 Z-234
$1 N3 G65 H2 X80 Z-200 Q4
...
  
```

4.12.2 Slide Synchronization

G functions can be used for synchronizing slides when more than one slide is used for machining a workpiece. The slides are synchronized by markers and/or tool positions in NC blocks which are started simultaneously.

One-sided synchronization

The slide programmed with G62 waits until "slide Q" has reached "mark H" or the mark **and** the X/Z coordinate. Use G162 to set a synchronizing mark for a different slide.

The CNC PILOT uses the **actual value**, if you synchronize to the X/Z coordinate.

Parameters

- H: Number of the marker (range: 0 <= H <= 15)
- Q: Slide to be awaited
- X, Z: Coordinate to end the waiting process – default: Only the marker is used for synchronizing

```
Example: G62
$1 N.. G62 Q2 H5 [slide 1 waits until slide 2 has
reached mark 5]
$2 N.. G62 Q1 H7 X200 [slide 2 waits until slide 1 reaches
mark 7 and the position X200]
...
```

Synchronous start of slides G63

G63 causes a **synchronous (simultaneous) start** of the programmed slides.

```
Example: G63
$1$2 N.. G63
$1 N.. G1 X.. Z.. [$1, $2 start simultaneously]
$2 N.. G1 X.. Z..
...
```


Synchronization marking G162

G162 sets a synchronizing mark. (Another slide programmed with G62 waits until the mark has been crossed.)


The slide continues executing the NC program without a pause.

Parameters

- H: Number of the mark (range: 0<=H<=15)



- Both slides must be controlled by the common main program.
- Do not synchronize to the end coordinates of NC blocks, because the positions to be used for synchronization must also be **crossed over** or are not reached due to servo lag. **Alternative:** Synchronous start of paths with G63
- Besides the G functions, the "program synchronization M97" is also available (see "4.17M Functions").



Ensure that no M or T commands are contained between the NC block containing G63 and the blocks containing traverse commands.

4.12.3 Spindle Synchronization, Workpiece Transfer

Spindle synchronization G720

G720 controls the workpiece transfer from the master to the slave spindle and synchronizes functions such as polygonal turning jobs.

Program the speed of the master spindle with Gx97 S.. and define the speed ratio between the master spindle and the slave spindle with "Q, F" If you enter a negative value for Q or F, the direction of rotation of the slave spindle will be reversed. If you synchronize more than one slave spindle with a master spindle, use G720 for each of the slave spindles.

The following rule applies: **Q * master speed = F * slave speed**

Parameters

- S: Number of the master spindle [1..4]
- H: Number of the slave spindle [1..4] – no input or H=0: Switches off the spindle synchronization
- C: Offset angle [°] – default: 0°
- Q: Master speed factor – default: 1;
Range: -100 <= Q <= 100
- F: Slave speed factor – default: Q is transferred;
range: -100 <= F <= 100

C-angle offset G905

G905 measures the "angular offset" of workpiece transfer with rotating spindle. The sum of angle C and the angle offset goes into effect as "zero point shift of C axis." This value is saved in the variable V922 (C axis 1) or V923 (C axis 2).

The zero offset stays in effect until another NC program is activated.

Parameters

- Q: Number of the C axis
- C: Angle of additional zero point shift for offset gripping – range: -360° <= C <= 360°; default: 0°

Measuring angular offset during spindle synchronization G906

G906 writes the angular offset between the master spindle and the slave spindle into variable V921.

Programming notes:

- Program G906 only for active angular synchronization – both chucks must be closed.
- Program G906 in a separate NC block.
- Program G909 (interpreter stop) **before processing** variable V921.
- G906 generates an "interpreter stop."

Example: G720

```
...
N.. G397 S1500 M3 [speed and direction
                        of master spindle]
N.. G720 C180 S4 H2 Q2 F-1 [synchronization
                        of master spindle to slave spindle. The slave
                        spindle precedes the master spindle
                        by 180°. Slave spindle: Direction of rotation M4;
                        speed 750]
...
```



Danger of collision!

- For narrow workpieces the jaws have to grip at an offset.
- The "zero point shift of C axis" is retained:
 - When changing from Automatic to Manual mode
 - After switch-off

Traversing to a fixed stop G916

G916 activates the “monitoring function for the traversing path.” Then you move with G1 to a “dead stop.” The CNC PILOT stops the slide as soon as the following error “lag” is reached, saves the position and retracts by the “reversing path” to reduce forces.

Application example

Transfer a premachined workpiece to a second traveling spindle if you do not know the exact position of the workpiece.

In machine parameters 1012, .. ;1112, 1162, .. define the following:

- Servo lag limit (to detect the dead stop)
- Reversing path

The CNC PILOT

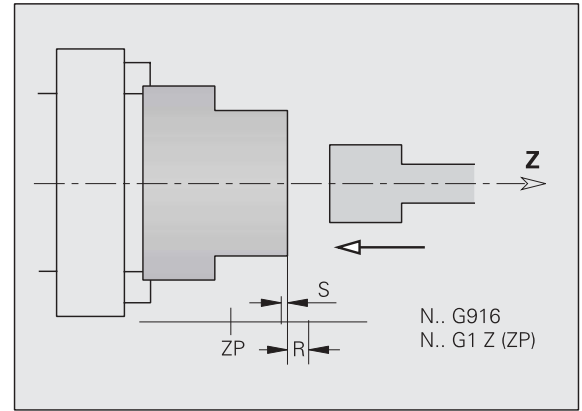
- Sets the feed rate override to 100%
- Moves to the dead stop and stops as soon as the servo lag error is reached—the path remaining is deleted
- Saves the dead stop position in the variables V901..V918
- Retracts by the “reversing path”
- Generates and “interpreter stop”

Programming notes:

- ▶ Position the slide at a sufficient distance before the stop
- ▶ Program G916 in the G1 positioning block
- ▶ Program G1 as follows:
 - Target position lies behind the dead stop
 - Move only **one** axis
 - “Feed per minute” must be active (G94)



As of software version 368 650-08, the traversing-to-a-dead-stop function can also be used for C axes.



ZP: Target position of the traverse command
S: Lag error limit
R: Reversing path

Example

```
...
N.. G94 F200
$2 N.. G0 Z20 [Pre-position slide 2]
$2 N.. G916 G1 Z-10 [Activate monitoring,
move to a dead stop]
...
```

Controlled parting using lag error monitoring G917

The controlled parting function (cut-off control) prevents collisions caused by incomplete parting processes. G917 monitors the path of traverse.

Application

■ Parting control

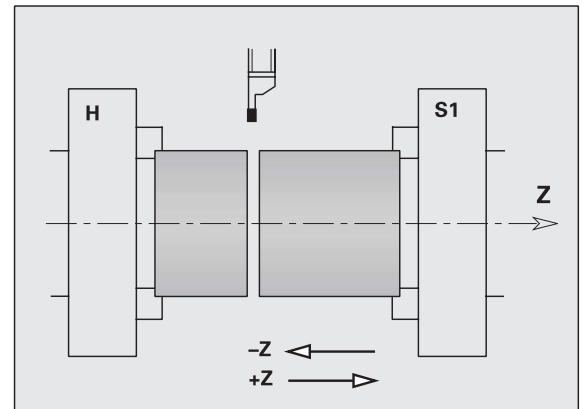
Move the workpiece in the positive Z direction after it has been cut off. If a lag error occurs, the workpiece is defined as follows:
No cut-off.

■ Smooth parting control

Move the workpiece in the negative Z direction after it has been cut off. If a lag error occurs, the workpiece is defined as follows:
No correct cut-off.

Define the following in machine parameters 1115, 1165, ...:

- Lag limit.
- Feed rate for the path of traverse to be monitored.



Continued ►

Programming controlled parting:

- ▶ Cut off the workpiece.
- ▶ Activate the monitoring function for the path of traverse with G917.
- ▶ Move the workpiece with G1 after it has been cut off.
- ▶ The CNC PILOT checks the lag error and stores the result in variable V300.
- ▶ Evaluate variable V300.

Test results

G917 produces satisfactory results provided that:

- With rough chucking jaws, $n < 3000$ rpm.
- With smooth chucking jaws, $n < 2000$ rpm.
- Clamping pressure of > 10 bar.

Controlled parting using spindle monitoring G991

The controlled parting function (cut-off control) prevents collisions caused by incomplete parting processes. G991 controls the parting process by monitoring the speed difference between the two spindles.

The spindles are connected in terms of actuation via the workpiece. After the parting process has been completed, the spindles rotate independently of each other. Speed differences and monitoring time are stored in machine parameters 808, 858, ... They can be changed with G992.

The CNC PILOT writes the monitoring result into variable V300.

Specify the path to be monitored in "return path R" and define whether the path before the workpiece is cut off or the return path is to be monitored (see illustration).

Parameters

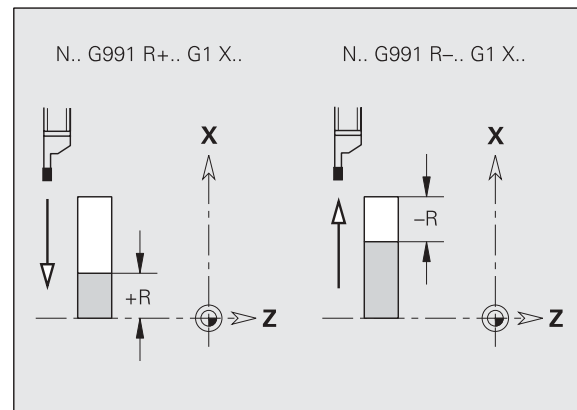
- R: Return path (radius value)
- No entry: The speed difference between the spindles running synchronously is checked (one-time check).
 - $R > 0$: "Remaining path before the parting operation" is monitored.
 - $R < 0$: "Return path" is monitored – the monitoring function is activated when the return path begins and is deactivated when the value "-R" is reached.



- Parting control with G917 is preferable to G991.
- Tool breakage might result in speed differences which in turn might affect the monitoring result. It is therefore advisable to monitor the reverse path too.

Programming notes:

- Program G917 and G1 in one block
- Program G1 as follows:
 - With "parting control": Path > 0.5 mm (to permit a result of monitoring)
 - When checking for smooth parting control: Path $<$ width of the parting tool
- Result in variable V300
 - 0: Workpiece has not been cut off correctly/smoothly (lag error has been detected).
 - 1: Workpiece has been cut off correctly/smoothly (no lag error has been detected).
- G917 generates an interpreter stop.



Programming notes:

- Program constant cutting speed G96.
- Program G991 and G1 (path before the parting operation or return path) in one block.
- Result stored in V300:
 - 0: Workpiece not cut off
 - 1: Workpiece cut off
- G991 generates an interpreter stop.

Values for controlled parting G992

G992 overwrites machine parameters 808, 858, ... for controlled parting.

The new parameters are effective from the next NC block and are retained until G992 is reprogrammed or the parameters are edited.

Parameters

- S: Speed difference (in revolutions per minute)
- E: Monitoring time (in ms)

4.12.4 Contour Follow-Up

The following G functions can be used to influence contour regeneration (see section "4.10.2 Contour Repetitions"). Examples: Program repeats (for the machining of bars), program branches, etc.

Storing/loading contour follow-up G702

Parameters

- Q: Loading/saving the contour
 - Q=0: Save – saves the current contour – no influence on the contour follow-up
 - Q=1: Load – loads the saved contour – the contour follow-up is resumed with the "loaded contour"



Program G702 for one slide only – usually, for slide 1.

Contour follow-up G703

With an IF, WHILE or SWITCH statement, the contour follow-up is switched off. With ENDIF, ENDWHILE or ENDSWITCH it is switched on again.

G703 switches the contour follow-up on for the THEN, ELSE or CASE branch.

Parameters

- Q: Contour regeneration On/Off
 - Q=0: Off
 - Q=1: On

K default branch G706

With IF or SWITCH statements with V variables, G706 defines the "default branch." The commands of the default branch are used for updating the "technology data" (tool, tool position, contour follow-up, TRC, etc.). After the branch, the result of the "default branch" goes into effect. Without the default branch, the technology data remain undefined after the branch.

Parameters

- Q: K branch
 - Q=0: No default branch defined;
 - Q=1: THEN branch as default branch
 - Q=2: ELSE branch as default branch
 - Q=3: Current branch as default branch

Programming notes:

Program:

- G706 Q0, 1, 2: before the branch
- G706 Q3: at the beginning of the THEN, ELSE or CASE branches

4.12.5 In-Process Measuring

Prerequisite:Touch trigger probe

The measuring results are evaluated by the NC program. You can use the **tool-life monitoring function** when the NC program sets "tool diagnosis bit 4 - tool wear determined by **in-process measuring of a workpiece**" in order to inform the CNC PILOT of a worn-out tool (see "4.2.4Tool Programming").

Switch on in-process measuring G910

G910 activates the probe and the probe monitoring function.

Programming notes:

- Program G910 alone in the NC block
- G910 is a modal function.
- G913 switches the touch probe off again

Actual value determination for in-process measurement G912

G912 stores the position of the probe in variablesV901 toV920 (see "4.15.2VVariables").

The CNC PILOT moves to the position to be measured and stops as soon as the probe stylus is deflected.The remaining path of traverse is deleted.You can influence the reaction to the "probe not triggered after moving the measuring path" situation with "error evaluation Q."

Parameters

- Q: Error evaluation – default: 0
- Q=0: "Cycle stop" condition; the error is displayed
 - Q=1: "Cycle on" condition; the error number 5518 is saved in variableV982

Switch off in-process measuring G913

G913 is used to deactivate probe monitoring. Before G913 becomes effective, retract the probe.

Program only G913 in the NC block. G903 generates an interpreter stop.

Switch off probe monitoring G914

After the deflection of the probe stylus, deactivate probe monitoring in order to retract the probe.

To retract the probe: Program G914 and G1 in an NC block

Note on programming in-process measuring:

- Position the probe at a sufficient distance before the "measuring point."
- Program G1 as follows:
 - Target position lies sufficiently behind the "measuring point"
 - Feed per minute must be active (G94)

Example: In-process measuring

...
BEARBEITUNG [MACHINING]
...
N..T .. [Change the touch probe]
N.. G910 [Activate in-process measuring]
N.. G0 .. [Pre-position probe]
N.. G912
N.. G1 .. [Approach probe]
N.. G914 G1 .. [Retract probe]
...
N.. G913 [Deactivate in-process measuring]
... [Evaluate measured values]



- X values are measured as a radius.
- The variables are also used by other G functions (G901, G902, G903 and G916). Make sure that your measuring results are not overwritten.

4.12.6 Post-Process Measuring

The workpieces are measured outside of the lathe and the results are transferred to the CNC PILOT. The measuring setup determines whether measured values or compensation values are transferred.

A **global measuring result** should be transferred to measuring position 0.

The results are evaluated by the NC program. Example: Tool wear compensation. You can use the **tool-life monitoring function** when the NC program sets "tool diagnosis bit 5 - tool wear determined by **in-process workpiece measuring**" in order to inform the CNC PILOT of a worn-out tool (see "4.2.4 Tool Programming").



The post-process measuring status as well as the measured values last received can be checked in Automatic mode (see "3.5.9 Post-process Measuring Status Display").

Post-process measuring G915

With G915, measured values are received from the post-process measuring system and stored in variables.

Variable assignment

- V939: Global measuring result
- V940 Measuring status
 - 0: **No** new measured values
 - 1: New measured values
- V941..V956 (corresponding to measuring points 1..16).

Parameters

- H: Block
- H=0: Reserved for further functions.
 - H=1: Measured values received are read in.



Evaluate the **measurement status** in order to avoid that compensation values are accounted for twice or incorrectly.

Example: Using measuring result for compensation		
...		
BEARBEITUNG		[MACHINING]
...		
N2T1		[Contour finishing - Outside]
...		
N49 ...		[End of machining process]
N50 G915 H1		[Request measuring results]
N51 IF {V940 == 1}		[If results are available]
N52 THEN		
N53 V {D1 [X] = D1 [X] + V941}		[Add the measuring result to compensation value D1]
N54 ENDIF		
...		

Example: Monitoring for tool breakage	
(Monitoring a limiting value)	
...	
BEARBEITUNG [MACHINING]	
...	
N2T1	[Contour Roughing - Outside]
...	
N49 ...	[End of machining process]
N50 G915 H1	[Request measuring results]
N51 IF {V940 == 1}	[If results are available]
N52 THEN	
N53 IF {V941 >= 1}	[Measured value > 1 mm]
N54 THEN	
N55 PRINTA ("Measured value > 1 mm = tool breakage")	
N56 M0	[Programmed stop – Cycle stop]
N57 ENDIF	
N58 ENDIF	
...	

4.12.7 Load Monitoring

The load monitoring function checks the performance and work values of the drives and compares them to limit values which have been determined during a **reference machining cycle**.

The CNC PILOT considers two limit values:

- If the first limit value is exceeded, the tool is marked as worn out and the **tool life monitoring** inserts the replacement tool during the next program run (see “4.2.4 Tool Programming”).
- Second limit value exceeded: The load monitor reports a broken tool and stops the program run (feed stop).

Defining a monitoring zone G995

G995 defines the monitoring zone and the axes to be monitored.

- G995 with parameter: Beginning of monitoring zone n
- G995 without parameter: End of the monitoring zone (not required, if another monitoring zone follows)

The zone number must be unambiguous in the NC program. A maximum of 49 monitoring zones per slide are possible.

Parameters

- H: Zone number—range: 1 to 999
- Q: Code for axes (drives to be monitored):
- 1: X axis
 - 2: Y axis
 - 4: Z axis
 - 8: Spindle
 - 16: Spindle 1
 - 128: C axis 1

Add the codes if you want to monitor more than one drive.
(Example: Monitoring the Z axis and main spindle: Q=12.)

Type of load monitoring G996

With G996 you can temporarily switch off the load monitoring and define the type of monitoring.

Parameters

- Q: Scope of monitoring—default: 0
- Q=0: Monitoring not active (effective for the entire NC program; even previously programmed G995 are rendered ineffective)
 - Q=1: Rapid traverse movements not monitored
 - Q=2: Rapid traverse movements monitored.
- H: Type of monitoring—default: 0
- H=0: Torque and work monitoring
 - H=1: Torque monitoring
 - H=2: Work monitoring.

Example: Load monitoring

```
...
MACHINING
...
N.. G996 Q1 H1    [Torque monitoring—does
                  not monitor rapid traverse paths. ]
...
N.. G14 Q0
N.. G26 S4000
N.. T2
N.. G995 H1 Q9    [Monitors the spindle
                  and X axis]
N.. G96 S230 G95 F0.35 M4
N.. M108
N.. G0 X106 Z4
N.. G47 P3
N.. G820 NS..    [Monitors the feed paths
                  of the roughing cycle]
N.. G0 X54
N.. G0 Z4
N.. M109
N.. G995          [Ends the machining cycle]
...
```



The “code for axes” is defined in bit numbers for load monitoring (control parameter 15).

4.13 Other G Functions

Period of dwell G4

The CNC PILOT interrupts the program run for the time F before executing the next program block. If G4 is programmed together with a path of traverse in the same block, the dwell time only becomes effective after the path of traverse has been executed.

Parameters

F: Period of dwell [sec] – range: $0 < F < 99.999$

Precision stop ON G7

G7 switches "precision stop" on. It is a modal function. With "precision stop" the CNC PILOT does not run the following block until the last point has been reached in the tolerance window for position (tolerance window: machine parameter 1106, ff "position control for linear axis").

"Precision stop" affects single contours and cycles. The NC block containing G7 is also executed with a precision stop.

Precision stop OFF G8

G8 switches "precision stop" off. The block containing G8 is executed **without** an accurate stop.

Block precision G9

G9 activates an accurate stop for the block in which it is programmed (see also "G7").

Move rotary axis G15

G15 is used to move the rotary axis to a specified angle. The principal and secondary axes can be moved on a linear path.

Parameters

A, B: Angle – end point of rotary axis

X, Y, Z: End point of principal axis (X diameter value)

U, V, W: End point of secondary axis



Programming of all parameters:

Absolute, incremental or modal.

Converting and mirroring G30

G30 converts G functions, M functions, and slide and spindle numbers with the aid of the conversion lists (machine parameters 135 and following). G30 mirrors traverse paths and tool dimensions and shifts the machine zero point about the “zero point offset” of the axis (see machine parameters 1114, 1164, ..).

Application:

For full-surface machining, you describe the complete contour, machine the front face, rechuck the workpiece (through an expert program) and then machine the rear face. To enable you to program rear-face machining in the same way as front-face machining (Z axis orientation, arc rotational direction, etc.). Includes the expert program commands for converting and mirroring.

Parameters

- H: Table number
- H=0: Switch off the conversion and compensate the offset
 - H=1..4: conversion table; in addition, the machine zero offset is activated (machine parameters 1114, 1164, and following).
- Q: Selection
- Q=0: Deactivate traverse path and tool mirroring
 - Q=1: Activate traverse path mirroring for specified axes
 - Q=2: Activate tool dimension mirroring for specified axes
- X, Y, Z, U, V, W, A, B, C—axis selection
- X=0: Mirroring of the X axis deactivated
 - X=1: Mirroring of the X axis activated
 - Y=0: Mirroring of the Y axis deactivated , etc.



- Mirror the traverse paths **and** tool lengths in separate G30 commands.
- Q1, Q2 without axis selection switches the mirroring function off.
- Only configured axes can be selected.



Danger of collision!

- In the transition from AUTOMATIC to MANUAL OPERATION, conversions and mirror images are retained.
- The conversion/mirroring must be switched off if you activate the front-face machining after rear-face machining (for example during program section repeats with M99).
- After a new program selection, the conversion/mirroring is switched off (example: transition from MANUAL to AUTOMATIC mode).

Switch off protection zone G60

G60 is used to cancel protective zone monitoring. G60 is programmed **before** the traversing command to be monitored or not monitored.

Application example:

With G60, you can temporarily deactivate a programmed monitoring of the protective zone in order to machine a centric through hole.

Parameters

- Q: ■ Q=0: Activate the protection zone (modal)
- Q=1: Deactivate the protection zone (modal)
 - Q no input: Deactivate the protection zone for the current NC block

Spindle with workpiece G98

The assignment of the spindle is required for thread-cutting, drilling, and milling cycles if the workpiece is not in the main spindle.

Parameters

- Q: Spindle number—default: 0 (master spindle)

Waiting for time G204

G204 interrupts an NC program up to a specified moment.

Parameters

D: Day (D=1..31) – default: First possible time "H, Q"
 H: Hour (H=0 to 23)
 Q: Minute (Q=0 to 59)

Update nominal values G717

G717 is used to update the nominal position values of the control with the axis position data.

Application:

- Deleting the lag error.
- Standardization of the slave axes after deactivating a master-slave coupling.

Move lag error G718

G718 prevents the automatic updating of nominal control position values with the axis position data (e.g. when traversing to a fixed stop, or after canceling and re-enabling a controller release).

Application:

Before activating a master-slave axis coupling.

Parameters

Q: On/Off
 ■ Q=0: Off
 ■ Q=1: On, the lag error is retained in the control memory.

Actual values to variables G901

G901 is used to transfer the actual values to the variables V901..V920 (see "4.15.2 Variables").

G903 generates an interpreter stop.

Zero-point shift to variables G902

Transfers the shift in **Z direction** into the variable V901..V920 (see "4.15.2 Variables").

G903 generates an interpreter stop.

Servo lag to variables G903

G903 transfers the current following error (distance by which the actual values lags the nominal value) into the variables V901..V920 (see "4.15.2 Variables").

G903 generates an interpreter stop.



Use **G717** and **G718** in expert programs only (refer to your commissioning manual for information on real time coupling function).

Block speed monitoring off G907

The CNC PILOT starts machining operations requiring spindle revolutions when the programmed speed has been reached. G907 is used to deactivate speed monitoring block by block - the path of traverse is started immediately.

Program G907 and the traverse path in the same NC block.

Feed override 100% G908

G908 sets the feed override for traverse paths (G0, G1, G2, G3, G12, G13) block by block to 100%.

Program G908 and the traverse path in the same NC block.

Interpreter stop G909

The CNC PILOT pre-interprets approx. 15 to 20 NC blocks in advance. If variables are assigned shortly before the evaluation, "old values" would be processed. An **interpreter stop** ensures that the variables contain the new value.

G909 stops the pre-interpretation. The NC blocks are processed up to G909 – after G909 the subsequent NC blocks are processed.

Apart from G909, the NC block should only contain synchronous functions. (Some G functions generate an interpreter stop.)

Look ahead G918

The look-ahead function can be activated/deactivated with G918. G918 can be programmed in a separate NC block program before/after the thread cutting (G31, G33).

Parameters

Q: Look-ahead function on/off – default: 1
 ■ Q=0: Off
 ■ Q=1: On

Spindle override 100% G919

Switched the spindle speed override off/on.

Parameters

Q: Spindle number – default: 0
 H: Type of limit – default: 0
 ■ H=0: Switch on spindle override
 ■ H=1: Spindle override to 100 % – modal
 ■ H=2: Spindle override to 100 % – for the current NC block

Deactivate zero offsets G920

"Deactivate" the workpiece zero point and the zero point shifts.
Traverse paths and position values are referenced to the distance **tool tip - machine zero point**.

Deactivate zero offsets, tool lengths G921

"Deactivates" the workpiece zero point, zero point shifts and tool dimensions. Traverse paths and position values are referenced to the distance **slide reference point – machine zero point**.

Lag error limit G975

Switches to "Lag error limit 2" (see machine parameter 1106, ..).

G975 is a modal function. At the end of a program the CNC PILOT switches to the standard lag error limit.

Parameters

Q: Lag error limit – default: 1
■ H=1: Standard lag error limit
■ H=2: Lag error limit 2

Activate zero offsets G980

"Activates" the workpiece zero point and all zero point shifts.

Traverse paths and position values are referenced to the distance **tool tip – workpiece zero point**, while taking the zero point shifts into consideration.

Activate zero offsets, tool lengths G981

"Activates" the workpiece zero point, all zero offsets and the tool dimensions.

Traverse paths and position values are referenced to the distance **tool tip – workpiece zero point**, while taking the zero point shifts into consideration.

4.14 Data Input and Data Output

Data are also entered and output during simulation. "V variables" are included during simulation. The V variables can be assigned values. Thus all branches of your NC program can be tested.

4.14.1 Input/Output of #Variables

INPUT

With INPUT you program the input of # variables that are evaluated during program interpretation.

You define the input text and the variable number. The CNC PILOT stops the interpretation at INPUT and waits for input of the variable value.

The CNC PILOT displays the input after having completed the INPUT command.

Syntax: INPUT ("text," variable)

PRINT

PRINT can be used to output texts and variable values during program interpretation. You can program a succession of several texts and # variables.

Syntax: PRINT ("text1,"variable,"text1,"variable, ..)

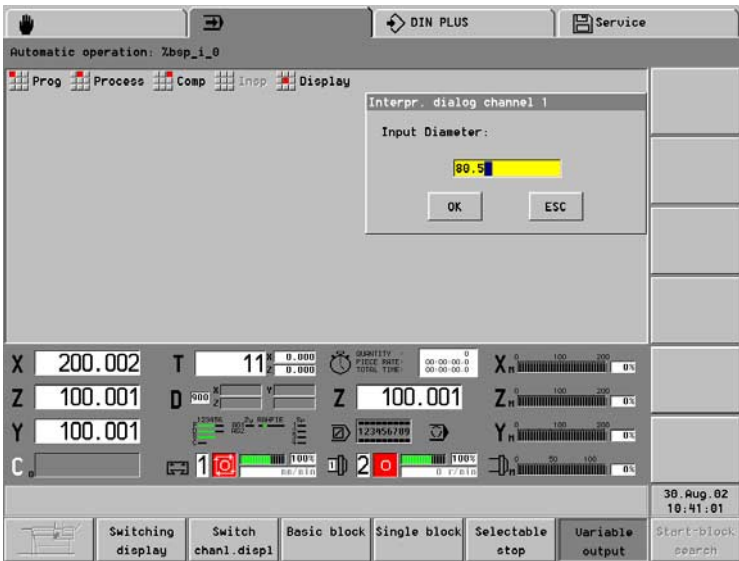
WINDOW

WINDOW (x) opens an output window with "x" lines. The window is opened as a result of the first input/output. WINDOW (0) closes the window.

The "standard window" comprises 3 lines – you do not need to program it.

Syntax:

WINDOW (number of lines) – 0 <= number or lines <= 10



Examples:

...

N.. WINDOW(8)

...

N.. INPUT("input diameter: ",#1)

...

N.. PRINT("input diameter: ",#1)

...

4.14.2 Input/Output of V Variables

INPUTA

“INPUTA” can be used to program the input of V variables evaluated during program run (run time).

You define the input text and the number of the variable. The CNC PILOT requests the input of the variable value during the execution of the command. The input is assigned to the variable and the program run continues.

The CNC PILOT displays the input after having completed the INPUT command.

Syntax: INPUTA(“Text”,variable)

PRINTA

“PRINTA” can be used to display text and V-variable values in an output window. You can program in succession up to two texts and up to two variables. The input must not exceed 80 characters.

To output texts and variable values on a printer, activate printer output in control parameter 1 (printer output ON).

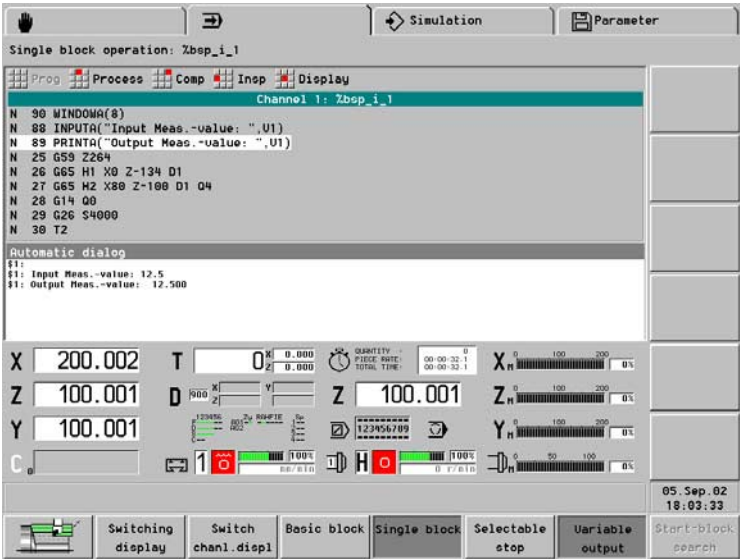
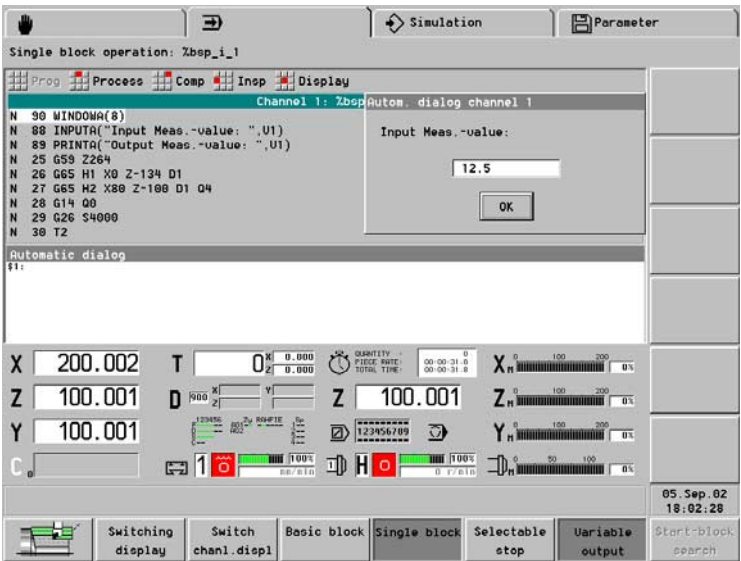
Syntax:
PRINTA(“Text1”,Variable,“Text1”,Variable, ..)

WINDOWA

WINDOWA (x) opens an output window with “x” lines. The window is opened as a result of the first input/output. WINDOWA (0) closes the window.

The “standard window” comprises 3 lines—you do not need to program it.

Syntax:
WINDOWA (line number) – 0 <= line number <= 10



Examples:

...

MACHINING

...

N100 WINDOWA(8)

...

N110 INPUTA(“Input Meas.-value:”,V1)

...

N120 PRINTA(“Output Meas.-value:”,V1)

...

4.15 ProgrammingVariables

The CNC PILOT supports NC programs before the program run. The system therefore differentiates between two types of variables:

- # variables are evaluated during **NC program interpretation**.
- V variables (or events) are evaluated during **NC program run**.

The following rules apply:

- Multiplication/division before addition/subtraction
- Up to 6 bracket levels
- **Integer variables** (only for V variables): Integer values between -32767 .. +32768
- **Real variable** (with # and V variable): Floating point numbers with max. 10 integers and 7 decimal places
- The variable are kept even if the control was switched off

4.15.1 #Variables

The CNC PILOT uses value ranges to define the **scope of variables**:

- **#0 .. #29: Channel-dependent, global variable**
Can be used for each slide (NC channel). Identical variable numbers on different slides are no problem.
Global variables are retained after the program has been completed and can be processed by the following NC program.
- **#30 .. #45 channel independent, global variable**
Are available once within the control. If the NC program of a slide changes a variable, it applies to all slides. The variables are retained after the program has been completed and can be processed by the following NC program.
- **#46 .. #50 variables are only used in expert programs**
. Do not use these variables in your NC program.
- **#256 .. #285 local variables**
are effective within a subprogram.

Reading-in parameter values

Syntax: **#1 = PARA(x,y,z)**

- x = Parameter group
 - 1: Machine parameters
 - 2: Control parameters
 - 3: Setup parameters
 - 4: Machining parameters
 - 5: PLC parameters
- y = Parameter number
- z = Sub-parameter number

Syntax	Mathematical function
+	Addition
–	Subtraction
*	Multiplication
/	Division
SQRT(...)	Square root
ABS(...)	Absolute amount
TAN(...)	Tangent (in degrees)
ATAN(...)	Arc tangent (in degrees)
SIN(...)	Sine (in degrees)
ASIN(...)	Arc sine (in degrees)
COS(...)	Cosine (in degrees)
ACOS(...)	Arc cosine (in degrees)
ROUND(...)	Round
LOGN(...)	Natural logarithm
EXP(...)	Exponential function e ^x
INT(...)	Cut decimal places

Only with **# variables**:

SQRTA(.., ..)	Square root of (a ² +b ²)
SQRTS(.., ..)	Square root of (a ² –b ²)



Program NC blocks containing variable calculations with "slide code \$.." if your lathe has more than one slide. Otherwise, the calculations are repeated.

Examples for "# variables"

...
N.. #1=PARA(1,7,3) [transfers "machine value 1 Z" to variable #1]
...
N.. #1=#1+1
N.. G1 X#1
N.. G1 X(SQRT(3*(SIN(30))))
N.. #1=(ABS(#2+0.5))
...

Continued ►

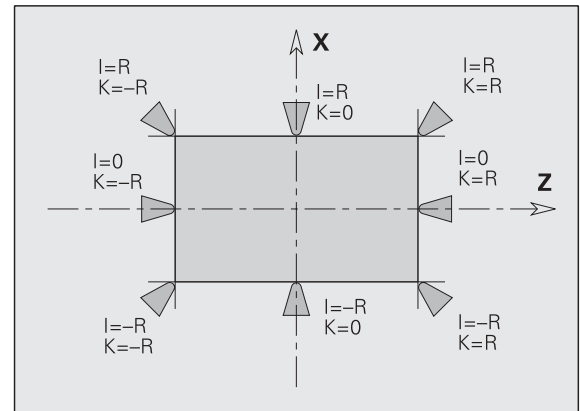
Information contained in variables

The following variable information on tool data and your NC program can be read out. The assignment of variables #518..#521 varies depending on the type of tool.

Precondition: The variable is "defined" as a result of a tool call or an NC program call.

# variables	Information on tool data
#512	Tool type (3-digit number)
#513..#515	1., 2., 3. Tool type position
#516	Usable length (nl) for turning, drilling or boring tools
#517	Main machining direction (see table)
#518	Secondary machining direction of turning tools (see table)
#519	Tool type: ■ 14*: 1 = right-hand model 2 = left-hand model (A) ■ 5**, 6**: number of teeth
#520	Tool type: ■ 1**, 2**: Cutting radius (rs) ■ 3**, 4**: Drilling/boring diameter (d1) ■ 51*, 52*: Milling diameter (front) (df) ■ 56*, 6**: Milling diameter (d1)
#521	Tool type: ■ 11*, 12*: Roughing/finishing diameter (sd) ■ 14*, 15*, 16*, 2**: Cutting width (sb) ■ 3**, 4**: Length of first cut (al) ■ 5**, 6**: Milling width (fb)
#522	Tool orientation (reference: machining direction of tool) 0: On the contour 1: To the right of the contour - 1: To the left of the contour
#523..#525	Set-up dimensions (ze, xe, ye)
#526..#527	Position of tool tip center I, K (see illustration)

# variables	NC information
#768..#770	Last programmed position X (radius value), Y, Z
#771	Last programmed position C [°]
#772	Active operating mode 2: Machine; 3: Simulation; 4: TURN PLUS
#774	Status TRC/MCRC 40: G40 active; 41: G41 active; 42: G42 active
#775	Number of the selected C axis



Positions and dimensions are always indicated in metric form. This also applies when an NC program is run in inches.

Primary and secondary machining direction:

0: undefined

1: + Z

2: + X

3: - Z

4: -X

5: +/- Z

6: +/- X

Continued ►

# variables	NC information
#776	Active wear compensation (G148) 0: DX, DZ; 1: DS, DZ; 2: DX, DS
#778	Unit of measure 0: Metric system; 1: Inch system
#782	Active machining plane 17: XY plane (front or back) 18: XZ plane (turning operation) 19: YZ plane (side view/surface)
#783, #785..#786	Distance from tool tip to slide reference point Y, Z, X
#787	Reference diameter for lateral surface machining (G120)
#788	Spindle holding the workpiece (G98)
#790	Oversize G52-Geo 0: Do not account for oversize 1: Account for oversize
#791..#792	G57 oversizes X, Z
#793	G58 oversize P
#794..#795	Cutting width in X, Z by which the tool reference point is shifted with G150/G151
#796	Number of spindle for which the last feed rate was programmed
#797	Number of spindle for which the last speed was programmed

4.15.2 V Variables

The CNC PILOT uses value ranges to define the scope of variables::

- Real V1 ..V199
- Integer V200 ..V299
- Reserved variables V300 ..V900

Requests and assignments:

■ **Machine dimensions** read/write (machine parameter 7)

Syntax: **V{Mx[y]}**

- x = dimension: 1..9
- y = coordinate: X,Y,Z,U,V,W,A,B or C

■ **Interrogate** external events

Is the **bit** value 0 or 1? The significance of the external event is determined by the machine manufacturer.

Syntax: **V{Ex[y]}**

- x = Slide 1..6
- y = bit: 1..16

Continued ►

■ **Interrogate**
The tool life monitoring function and the function for searching the start block trigger sequential events (see below).

Syntax: V{Ex[1]}
x = event: 20..59, 90
■ 20: Tool life has expired (global information)
■ 21..59: Tool life of **this** tool has expired
■ 90: Search for start block (0=not active; 1=active)
Assign this clock event to the tool ("tool life management"—Manual control mode).

■ **Tool compensation** Read/write

Syntax: V{Dx[y]}
x = T number
y = length compensation: X, Y, or Z

■ **Diagnosis bits** (Tool life monitoring) read/write

Syntax: V{Tx[y]}
x = T number
y = bit: 1..16 (see table)

Sequential events and tool life management

When a tool is worn-out, "event 20" (global information") and "event 1" are triggered. "Event 1" can be used to identify the worn-out tool. When the last tool of an tool interchange chain is worn out, "event 2" is also triggered.

Define "events 1 and 2" for each individual tool of the tool interchange chain.

The sequential events are automatically reset at the end of a program (M99).

Information in variables

- V660: Quantity
 - Is set to zero when the system is started.
 - Is set to zero when a **new** NC program is loaded
 - Is increased by M30 or M99 by 1
- V901..V920 are used for the G functions G901, G902, G903, G912 and G916 (see table).



If an tool interchange chain is defined, program the first tool of the chain for tool compensation and tool diagnosis. The CNC PILOT addresses the **active tool from the sequence of exchange** (see "4.2.4 Tool programming")

Example for diagnosis bits

. . .
N..V{T10[1]=1} [Sets the expiration of tool life of
tool 10—or replacement tool]
...

Tool diagnosis bits

Bit	Meaning
1	Worn-out tool—identifies tool condition. Reason for replacing a tool: see bits 2 to 8.
2	Specified tool life/piece number has been reached.
3	Reserved for "tool wear determined by in-process measuring of tool."
4	Tool wear determined by in-process measuring of workpiece.
5	Tool wear determined by in-process measuring of workpiece.
6	Tool wear, identified by the load monitoring function (power limit value 1 or 2 has been exceeded)
7	Tool wear, identified by the load monitoring function (power limit value 1 or 2 has been exceeded)
8	A cutting edge of a multipoint tool is worn-out.
9	New cutting edge
12	The remaining tool life of the cutting edge is <6% or the remaining piece number is 1.

- Bit=0: No; Bit=1: Yes
- Bits 9 to 16 are for general information.

Continued ►

- V921: Angular offset of “G906 spindle synchronization”
- V922/V923: Result of “G905 offset C-angle”
- V982: Error number of “G912 actual value determination for in-process measurement”
- V300: Result of “G991 controlled parting”

Examples for “V variables”	
...	
N..V{M1[Z]=300}	[Sets “machine dimension 1 Z” to “300”]
...	
N.. G0 Z{M1[Z]}	[Moves to “machine dimension 1 Z”]
...	
N.. IF{E1[1]==0}	[Interrogates “external event 1 – bit 1”]
...	
N..V{D5[X]=1.3}	[Sets “compensation X for tool 5”]
...	
N..V{V12=17.4}	
N..V{V12=V12+1}	
N.. G1 X{V12}	
...	


Note on interpreter stop (G909)
 The CNC PILOT pre-interprets approx. 15 to 20 NC blocks. If variables are assigned shortly before the evaluation, “old values” would be processed. An **interpreter stop** ensures that the variables contain the new value.

G909 stops the pre-interpretation. The NC blocks are processed up to G909—after G909 the subsequent NC blocks are processed.


4.15.3 Program Branches, Program Repeats, Conditional Block Execution

V variables are included during simulation. The V variables can be assigned values. Thus all branches of your NC program can be tested.

You can combine up to two conditions.




IF you program branches on the basis of V variables, there must not be any # variables in the program branches.



- The unit counter in V660 is different from the unit counter in the machine display.
- X values are saved as radius values.
- Note: Functions G901, G902, G903, G912 and G916 overwrite the variable! This also applies to variables that have not yet been evaluated.

Variable assignment V901 to V920			
	X	Z	Y
Slide 1	V901	V902	V903
Slide 2	V904	V905	V906
Slide 3	V907	V908	V909
Slide 4	V910	V911	V912
Slide 5	V913	V914	V915
Slide 6	V916	V917	V918
C axis 1:	V919		
C axis 2:	V920		



- Program an **interpreter stop** if variables or external events are modified shortly before the block is run.
- Each interpreter stop lengthens the run time of the NC program.
- Several G functions include the interpreter stop.

Logical operators for IF.. and WHILE..	
<	Less than
<=	Less than or equal
<>	Not equal
>	Greater than
>=	Greater than or equal
==	Equal

Combining conditions:	
AND	Logical AND operation
OR	Logical OR operation

IF..THEN..ELSE..ENDIF – program branches

A "conditional branch" consists of the elements:

- IF – followed by a condition. In the case of conditions, variables or mathematical expressions are defined to the left and to the right of relational operators.
- THEN – when the condition is fulfilled, the THEN branch is executed.
- ELSE – when the condition is not fulfilled, the ELSE branch is executed.
- ENDIF – concludes the conditional program branch.

Programming notes

- ▶ Select IF (Machining menu: Instructions – DIN PLUS words)
- ▶ Enter the desired condition (only enter the required brackets).
- ▶ Insert NC blocks of the THEN and ELSE branch – the ELSE branch can be omitted



■ NC blocks with IF, THEN, ELSE, ENDIF can have no further commands

■ For branches due to V variables or events, the contour follow-up is switched off with the IF statement and back on with the ENDIF statement. With G703 you can switch on the contour follow-up function.

Example:

```

. . .
N.. IF {E1[16]==1}
N.. THEN
N..   G0 X100 Z100
N.. ELSE
N..   G0 X0 Z0
N.. ENDIF
. . .

```

WHILE..ENDWHILE – program repeat

A "program repeat" consists of the elements:

- WHILE – followed by a condition. In the case of "conditions", variables or mathematical expressions are defined to the left and to the right of relational operators.
- ENDWHILE – concludes the conditional program branch.

NC blocks programmed between WHILE and ENDWHILE are executed repeatedly for as long as the "condition" is fulfilled. If the condition is not fulfilled, CNC PILOT continues execution of the program with the block programmed after ENDWHILE.

Programming notes

- ▶ Select WHILE (Machining menu: Instructions – DIN PLUS words)
- ▶ Enter the desired condition (enter only the required brackets).
- ▶ Insert NC blocks



■ If the repetition is made due to V variables or events, the contour follow-up function is switched off for the WHILE branch and switched on again with ENDWHILE. With G703 you can switch on the contour follow-up function.

■ If the "condition" you program in the WHILE command is always true, the program remains in an "endless loop." This is one of the most frequent causes of error when working with program repeats.

Example:

```

. . .
N.. WHILE (#4<10) AND (#5>=0)
N..   G0 Xi10
. . .
N.. ENDWHILE
. . .

```

SWITCH..CASE—Program branch

The “switch statement” consists of the elements:

- SWITCH—Followed by a variable. The content of the variable is interrogated in the following CASE statement.
- CASE x—This CASE branch is run with the variable value x. CASE can be programmed more than once.
- DEFAULT—This branch is run if no CASE statement matched the variable. DEFAULT can be omitted.
- BREAK—Closes the CASE or DEFAULT branches.

Programming notes

- Select SWITCH (Machining menu: Instructions—DIN PLUS words)
- Enter the variable (without parentheses)
- For each CASE branch:
 - Select CASE (Machining menu: Instructions—DIN PLUS words)
 - Enter the “SWITCH condition” (value of the variable).
 - Insert the NC blocks to be executed.
- For the DEFAULT branch:
 - Insert the NC blocks to be executed.

Skip level/..

An NC block with preceding skip level is **not run** with an **active** skip level (see “4.3.3 Machining Menu”).

Skip levels are activated/deactivated in Automatic mode (machine mode of operation).

In addition, you can use the **skip cycle** (setup parameter 11 “deletion level/cycle”). “Skip cycle x” activates the skip level every xth time.

Example: **/1 N 100 G...**

“N100” is **not** executed if skip level 1 is active.

Slide code \$..

An NC block preceded by a slide code is executed only for the indicated slide (see “4.3.3 Machining Menu”). NC blocks that are not preceded by slide codes are executed on all slides.



- If the branch is triggered by V variables or events, the contour follow-up is switched off with the SWITCH statement and back on with the ENDSWITCH statement. With G703 you can switch on the contour follow-up function.
- The variable value should be a whole number—it is not rounded.

Example:

```
N.. SWITCH {V1}
N..  CASE 1      [executed if V1=1]
N..  G0 Xi10
. . .
N..  BREAK
N..  CASE 2      [executed if V1=2]
N..  G0 Xi10
. . .
N..  BREAK
N..  DEFAULT    [executed if no]
N..  G0 Xi10     CASE instruction
. . .           matched the variable value]
N..  BREAK
N.. ENDSWITCH
. . .
```



- For lathes equipped with one slide or program heads containing only **one** slide, a slide code is not necessary.

4.16 Subprograms

Calling a subprogram: L"xx"V1

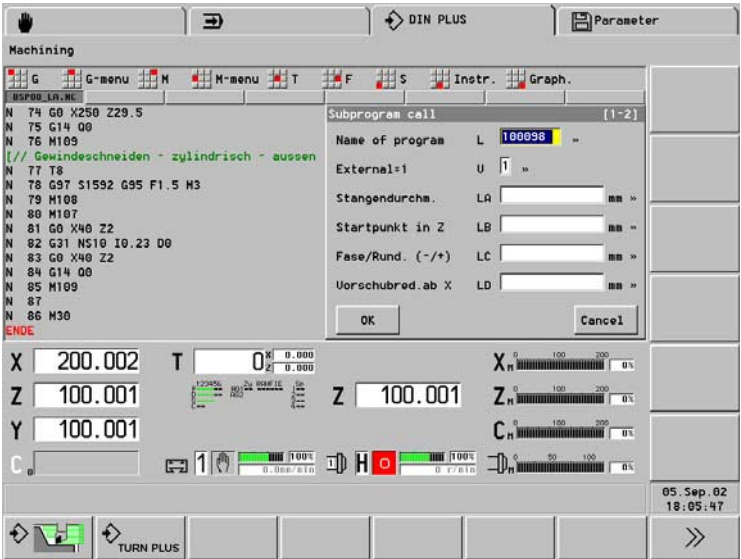
- L: Indicator of subprogram call
- "xx": Name of the subprogram – file name for external subprograms (max. 8 characters or letters)
- V1: Identification code for **external** subprograms – omitted for local subprograms

Note on using subprograms:

- External subprograms are defined in a separate file. They can be called from any main program, other subprograms, or from TURN PLUS.
- Local subprograms are in the main program file. They can be called only from the main program.
- Subprograms can be "nested" up to 6 times. Nesting means that another subprogram is called from within a subprogram.
- Recursion should be avoided.
- You can add up to 20 "transfer values" to a subprogram. The designations (parameter designations) are:
LA..LF, LH, I, J, K, O, P, R, S, U, W, X, Y, Z.
The transfer values are available as variables within the subprogram. The identification code is: "#__.", followed by the parameter designation in lowercase letters (for example: #__la).
You can use the transfer values when programming with variables within the subprogram.
- The variables #256..#285 are available in every subprogram as local variables.
- If a subprogram is to be executed repeatedly, define in the "number repeats Q" parameter the number of times the subprogram is to be repeated.
- A subprogram ends with RETURN.

Dialog texts

You can define the parameter descriptions that precede/follow the input fields in an external subprogram.
The CNC PILOT automatically sets the unit of measure for parameter values to the metric system or inches.
Max. 19 descriptions – the parameter descriptions can be positioned within the subprogram as desired.



The parameter "LN" is reserved for the transfer of block numbers. This parameter may receive a new value when the NC program is renumbered.

Parameter descriptions:

[/] – Beginning

[pn=n; s=parameter text (max. 16 characters)]

[/] – End

- pn: Parameter designations (la, lb, ...)
n: Conversion number for units of measurement
- 0: Non-dimensional
 - 1: "mm" or "inch"
 - 2: "mm/rev" or "inch/rev"
 - 3: "mm/min" or "inch/min"
 - 4: "m/min" or "feet/min"
 - 5: "Rev/min"
 - 6: Degrees (°)
 - 7: "µm" or "µinch"

Example

...

[/]

[la=1; s=bar diameter]

[lb=1; s=starting point in Z]

[lc=1; s=chamfer/round. (-/+)]

...

[/]

...

4.17 M Functions

M functions control the program run and the machine components (machine commands).

M00 Program STOP

The program run stops – “Cycle Start” resumes the program run.

M01 Optional STOP

The “optional stop” (Automatic mode) determines whether the program run stops at M01. “Cycle start” resumes the program run.

M30 End of program

M30 indicates the end of a program or subprogram. (M30 does not need to be programmed.)

If you press “Cycle START” after M30, program execution is repeated from the start of the program.

M99 Program end with restart at beginning of program or at a given block number

M99 means “end program and start again.” CNC PILOT restarts program execution from:

- Program beginning, if NS is not entered
- Block number NS, if NS is entered



Modal functions (feed rate, spindle speed, tool number, etc.) which are effective at the end of program remain in effect when the program is restarted. You should therefore reprogram the modal functions at the start of program or at the startup block (if M99 is used).

M97 Synchronous function

Slides for which M97 is programmed wait until **all slides** have reached this sentence. Program run then continues.

For complex machining operations (e.g. machining of several workpieces), M97 can be programmed with parameters.

Parameters

- H: Synchronous mark number – the evaluation takes place only during interpretation of the NC programs
- Q: Slide number – use synchronization with Q if synchronization with \$x is not possible
- D: On/Off – default: 0
- 0: Off – synchronization during runtime of NC program
 - 1: On – synchronization exclusively during interpretation of the NC programs

Example for M97

. . .

\$1 N.. G1 X.. Z..

\$2 N.. G1 X.. Z..

\$1\$2 N.. M97 [\$1, \$2 wait for each other]

. . .

Machine commands

The effect of the machine commands depends on the type of lathe. The table below lists the M commands used on most machines.



For more information on the M commands, refer to your machine manual.

M commands for program-run control

M00	Program STOP
M01	Optional STOP
M30	End of program
M99 NS..	Program end with restart

M commands as machine commands

M03	Spindle ON (CW)
M04	Spindle ON (CCW)
M05	Spindle STOP
M12	Lock spindle brake
M13	Release spindle brake
M14	C axis ON
M15	C axis OFF
M19 C..	STOP spindle at position “C”
M40	Shift gear to range 0 (neutral)
M41	Shift gear to range 1
M42	Shift gear to range 2
M43	Shift gear to range 3
M44	Shift gear to range 4
Mx03	Spindle x ON (CW)
Mx04	Spindle x ON (CCW)
Mx05	Spindle x STOP
M97	Synchronous function

4.18 Programming Notes and Examples

4.18.1 Programming Machining Cycles

PROGRAMMKOPF [PROGRAM HEAD]	Example: Typical structure of a machining cycle
...	
ROHTEIL [BLANK]	
...	
FERTIGTEIL [FINISHED PART]	
...	
BEARBEITUNG [MACHINING]	
N.. G59 Z..	Zero-point shift
N.. G26 S..	Definition of speed limit
N.. G14 Q..	Approach to tool change position.
. . .	
N.. T..	Tool change
N.. G96 S.. G95 F.. M4	Technology data: Cutting speed
	(spindle speed); feed rate; direction of rotation
N.. G0 X.. Z..	Positioning
N.. G47 P..	Definition of safety clearance
N.. G810 NS.. NE..	Cycle call
N.. G0 X.. Z..	If necessary: Retract
N.. G14 Q0	Approach to tool change position.
. . .	

4.18.2 Contour Repetitions

%111.nc	Example: Programming contour repetitions, including storing of the contour
PROGRAMMKOPF [PROGRAM HEAD]	
#SCHLITTEN \$1 [SLIDE]	
REVOLVER 1 [TURRET]	
T2 ID"121-55-040.1"	
T3 ID"111-55-080.1"	
T4 ID"161-400.2"	
T8 ID"342-18.0-70"	
T12 ID"112-12-050.1"	
SPANNMITTEL 1 [CHUCKING EQUIPMENT]	
ROHTEIL [BLANK]	
N1 G20 X70 Z120 K1	

FERTIGTEIL [FINISHED PART]	
N2 G0 X19.2 Z-10	
N3 G1 Z-8.5 B0.35	
N4 G1 X38 B3	
N5 G1 Z-3.05 B0.2	
N6 G1 X42 B0.5	
N7 G1 Z0 B0.2	
N8 G1 X66 B0.5	
N9 G1 Z-10 B0.5	
N10 G1 X19.2 B0.5	
BEARBEITUNG [MACHINING]	
N11 G26 S2500	
N12 G14 Q0	
N13 G702 Q0	Store contour
N14 L"1" V0 Q2	"Qx" = number of repetitions
N15 M30	
UNTERPROGRAMM "1" [SUBPROGRAM]	
N16 M108	
N17 G702 Q1	Load stored contour
N18 G14 Q0	
N19 T8	
N20 G97 S2000 M3	
N21 G95 F0.2	
N22 G0 X0 Z4	
N23 G147 K1	
N24 G74 Z-15 P72 I8 B20 J36 E0.1 K0	
N25 G14 Q0	
N26 T3	
N27 G96 S300 G95 F0.35 M4	
N28 G0 X72 Z2	
N29 G820 NS8 NE8 P2 K0.2 W270 V3	
N30 G14 Q0	
N31 T12	
N32 G96 S250 G95 F0.22	
N33 G810 NS7 NE3 P2 I0.2 K0.1 Z-12 H0	
W180 Q0	
N34 G14 Q2	
N35 T2	
N36 G96 S300 G95 F0.08	
N37 G0 X69 Z2	
N38 G47 P1	
N39 G890 NS8 V3 H3 Z-40 D3	

N40	G47	P1	
N41	G890	NS9 V1 H0 Z-40 D1 I74 K0	
N42	G14	Q0	
N43	T12		
N44	G0	X44 Z2	
N45	G890	NS7 NE3	
N46	G14	Q2	
N47	T4		Insert recessing tool
N48	G96	S160 G95 F0.18 M4	
N49	G0	X72 Z-14	
N50	G150		Shift reference point to the right of the cutting edge
N51	G1	X60	
N52	G1	X72	
N53	G0	Z-9	
N54	G1	X66 G95 F0.18	
N55	G42		Activate TRC
N56	G1	Z-10 B0.5	
N57	G1	X17	
N58	G0	X72	
N59	G0	X80 Z-10 G40	Switch off TRC
N60	G14	Q0	
N61	G56	Z-14.4	Incremental zero offset
RETURN			
ENDE [END]			

4.18.3 Full-Surface Machining

The term "full-surface machining" refers to the machining of the front and rear ends in **one** NC program. The CNC PILOT supports full-surface machining for all common machine designs. The features include angle-synchronous part transfer with rotating spindle, traversing to a stop, controlled parting, and coordinate transformation. This ensures efficient full-surface machining and simple programming.

You describe the turning contour, the contours for the C axis (or Y axis) as well as full-surface machining functions in one NC program. Expert programs are available for configuring the lathe. You can also use full-surface machining functions for single-spindle lathes.

Fundamentals

Rear-face contours with C axis: The XK axis and therefore also the C axis are oriented relative to the workpiece, not to the spindle.

Therefore, for the rear face:

- Orientation of the XK axis: "To the left" (front end: "to the right")
- Orientation of the C axis: "Clockwise"
- Rotational direction for arcs, G102: "Counterclockwise"
- Rotational direction for arcs, G103: "Clockwise"

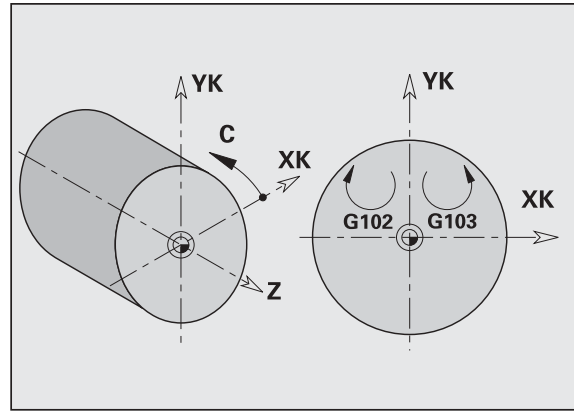
Rear-face contours with Y axis: The X axis is also oriented relative to the workpiece. Therefore, for the rear face:

- Orientation of the X axis: "To the left" (front end: "to the right").
- Rotational direction for arcs, G2: "Counterclockwise."
- Rotational direction for arcs, G3: "Clockwise."

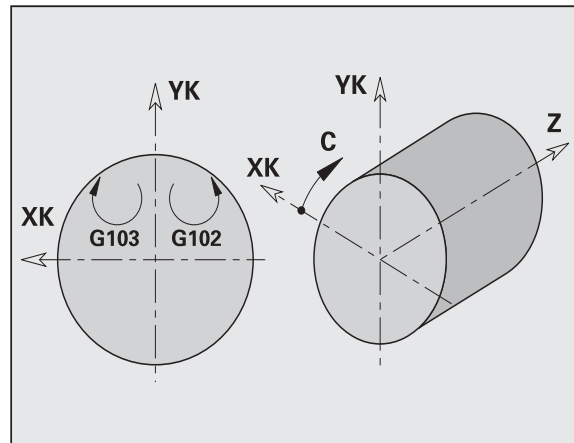
Turning: The CNC PILOT supports full-surface machining with conversion and mirroring functions so that rear-face machining also follows the principle:

- Program a **positive value** to depart the workpiece.
- Program a **negative value** to approach the workpiece.

The machine manufacturer usually supplies your lathe with suitable **expert programs** for workpiece transfer.



Front face



Rear face

Programming

When programming a contour on the rear face, be sure to consider the orientation of the XK axis (or X axis) and rotational direction of arcs.

Insofar as you use drilling and milling cycles, there are no special aspects to rear-face machining, since these cycles refer to predefined contours.

For rear-face machining with the basic commands G100..G103 (or G0..G3, G12.. G13 for the Y axis), the same conditions as for rear-face contours apply.

Turning

The expert programs contain converting and mirroring functions. The following principle applies for rear-face machining (2nd setup):

- **+ direction:** Program a positive value to depart the workpiece
- **- direction:** Program a negative value to approach the workpiece
- **G2/G12:** Circular arc "clockwise"
- **G3/G13:** Circular arc "counterclockwise"

Full-surface machining with counterspindle

G30: The expert program activates the mirroring of the Z axis and the conversion of the arcs (G2, G3, ..). The arcs must be converted for turning operations and machining with the C axis.

G121: The expert program shifts the contour and mirrors the coordinate system (Z axis). Further programming of G121 is normally not required for machining the rear face after rechucking.

Full-surface machining with single spindle

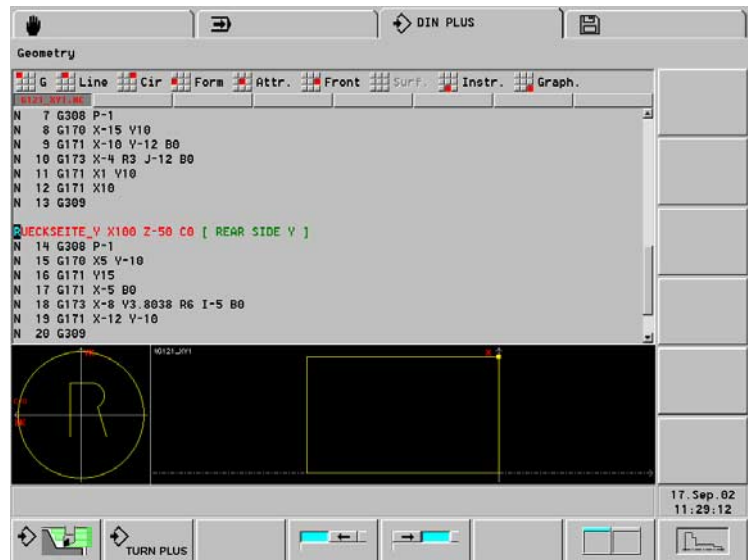
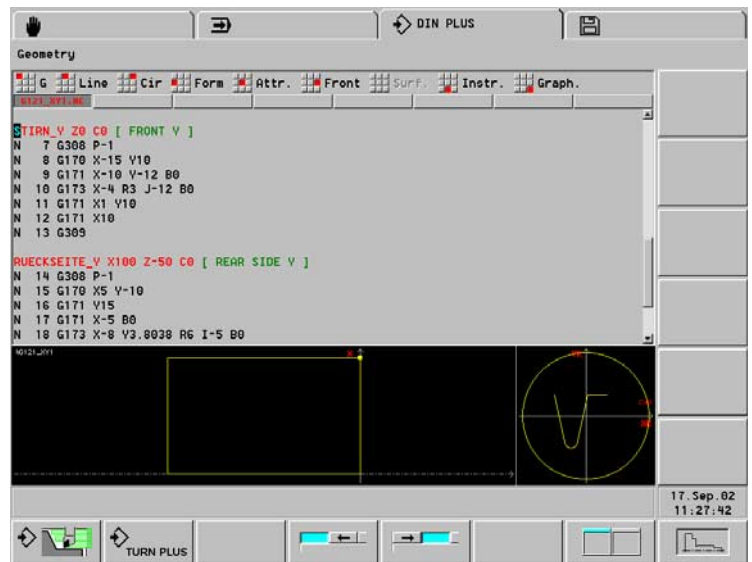
G30: Normally not required

G121: The expert program mirrors the contour. Further programming of G121 is normally not required for machining the rear face after rechucking.

Working without expert programs

If you do not use the expert programs or the converting and mirroring functions, the following principle applies:

- **+ direction:** Program a positive value to move away from the spindle
- **- direction:** Program a negative value to approach the spindle
- **G2/G12:** Circular arc "clockwise"
- **G3/G13:** Circular arc "counterclockwise"



For machining the rear face with the Y axis, deactivate arc conversion (G30 H2) and reactivate it for turning and other operations in the YZ plane (surface view) (G30 H1).

Example: Full-surface machining on lathe with traveling opposing spindle

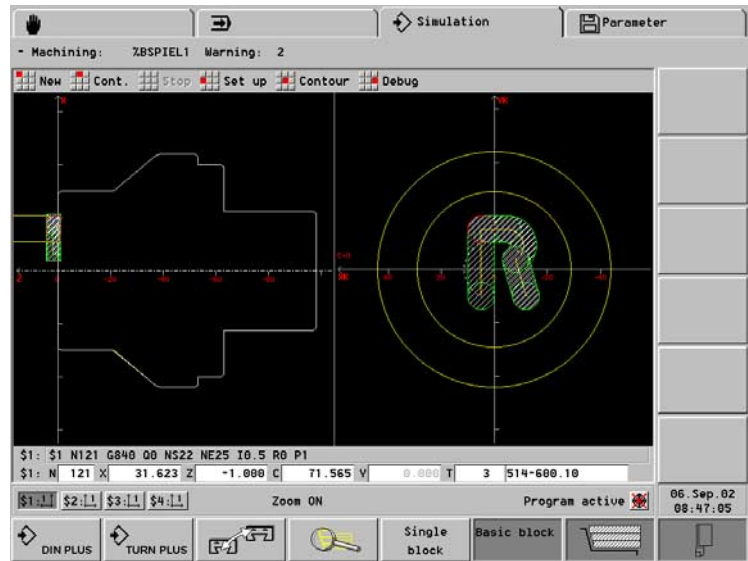
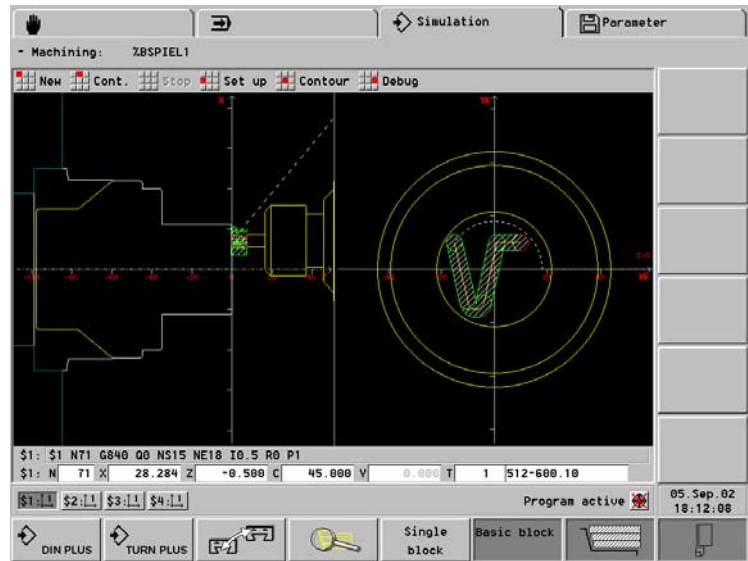
The workpiece is machined on the front face, transferred to the opposing spindle through an expert program and machined on the rear face.

- Upper figure: Machining the front face
- Lower figure: Machining the rear face.

The expert program is used for:

- Angular-synchronous workpiece transfer to counterspindle
- Mirroring the traverse paths for the Z axis
- Activating the conversion list
- Mirroring the contour definition and shifting it for the 2nd setup

The mirroring/conversion function for rear-face machining (expert program), is switched off at program end with the G30 command.



%bspie11.nc	Example: Full-surface machining on single-spindle counterspindle
PROGRAMMKOPF [PROGRAM HEAD]	
#SCHLITTEN \$1\$2 [SLIDE]	
. . .	
REVOLVER 1 [TURRET]	
T1 ID"512-600.10"	
T2 ID"111-80-080.1"	
T3 ID"514-600.10"	
T4 ID"121-55-040.1"	
T6 ID"115-80-080.1"	
T8 ID"125-55-040.1"	
SPANNMITTEL 1 [CHUCKING EQUIPMENT] [zero-point shift Z233]	Chucking equipment for 1st set-up
H1 ID"3BACK"	
H2 ID"KBA250-86" X100 Q4.	
SPANNMITTEL 4 [CHUCKING EQUIPMENT] [Zero-point shift Z196]	Chucking equipment for 2nd set-up
H1 ID"3BACK"	
H2 ID"WBA240-50" X80 Q4.	
ROHTEIL [BLANK]	
N1 G20 X100 Z100 K1	
FERTIGTEIL [FINISHED PART]	
. . .	
STIRN Z0 [FRONT]	
N13 G308 P-1	
N14 G100 XK-15 YK10	
N15 G101 XK-10 YK-12 B0	
N16 G103 XK-4.0725 YK-12.6555 R3 J-12	
N17 G101 XK1 YK10	
N18 G101 XK10	
N19 G309	
RUECKSEITE Z-98 [REAR SIDE]	
. . .	

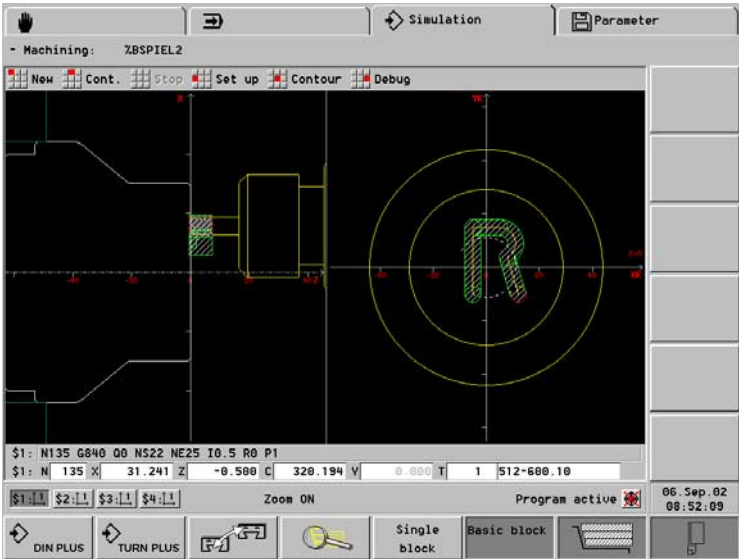
BEARBEITUNG [MACHINING]	
N27 G59 Z233	Zero-point shift 1st setup
\$1 N28 G65 H1 X0 Z-135 D1	Display chucking equipment of 1st setup
\$1 N29 G65 H2 X100 Z-99 D1 Q4	
\$1 N30 G14 Q0	
\$1 N31 G26 S2500	
\$1 N32 T2	
. . .	
\$1 N62 G126 S4000	Milling - Contour - Outside - Front face
\$1 N63 M5	
\$1 N64 T1	
\$1 N65 G197 S1485 G193 F0.05 M103	
\$1 N66 M14	
\$1 N67 M107	
\$1 N68 G0 X36.0555 Z3	
\$1 N69 G110 C146.31	
\$1 N70 G147 I2 K2	
\$1 N71 G840 Q0 NS15 NE18 I0.5 R0 P1	
\$1 N72 G0 X31.241 Z3	
\$1 N73 G14 Q0	
\$1 N74 M105	
\$1 N75 M109	
\$1 N76 M15	Prepare the rechucking
\$1 N77 G65 H1 D1	Delete chucking equipment of 1st set-up
\$1 N78 G65 H2 D1	
\$1 \$2 N79 M97	Synchronize slides for rechucking procedure
\$1 \$2 N80 L"UMKOMPL" V1 LA1000 LD369 LE547 LF98 LH98	Expert program for parting and rechucking
I3	LA=speed limitation
	LD=transfer position Z
	LE=machining position Z – slide 2
	LF=length of finished part
	LH=distance between chuck reference point
	and edge of workpiece
	I=minimum feed motion to fixed stop
\$1 \$2 N81 M97	
\$1 N82 G65 H1 X0 Z-100 D4	Activate chucking equipment for spindle 4
\$1 N83 G65 H2 X80 Z-63 D4 Q4	
. . .	Rear-face machining
\$1 \$2 N125 G30 H0 Q0	Deactivate rear-face machining
\$1 \$2 N126 M97	
N129 M30	
ENDE [END]	

Example: Full-surface machining on a single-spindle lathe

In the example, the machining of the front and rear face, using **one** NC program, is described.

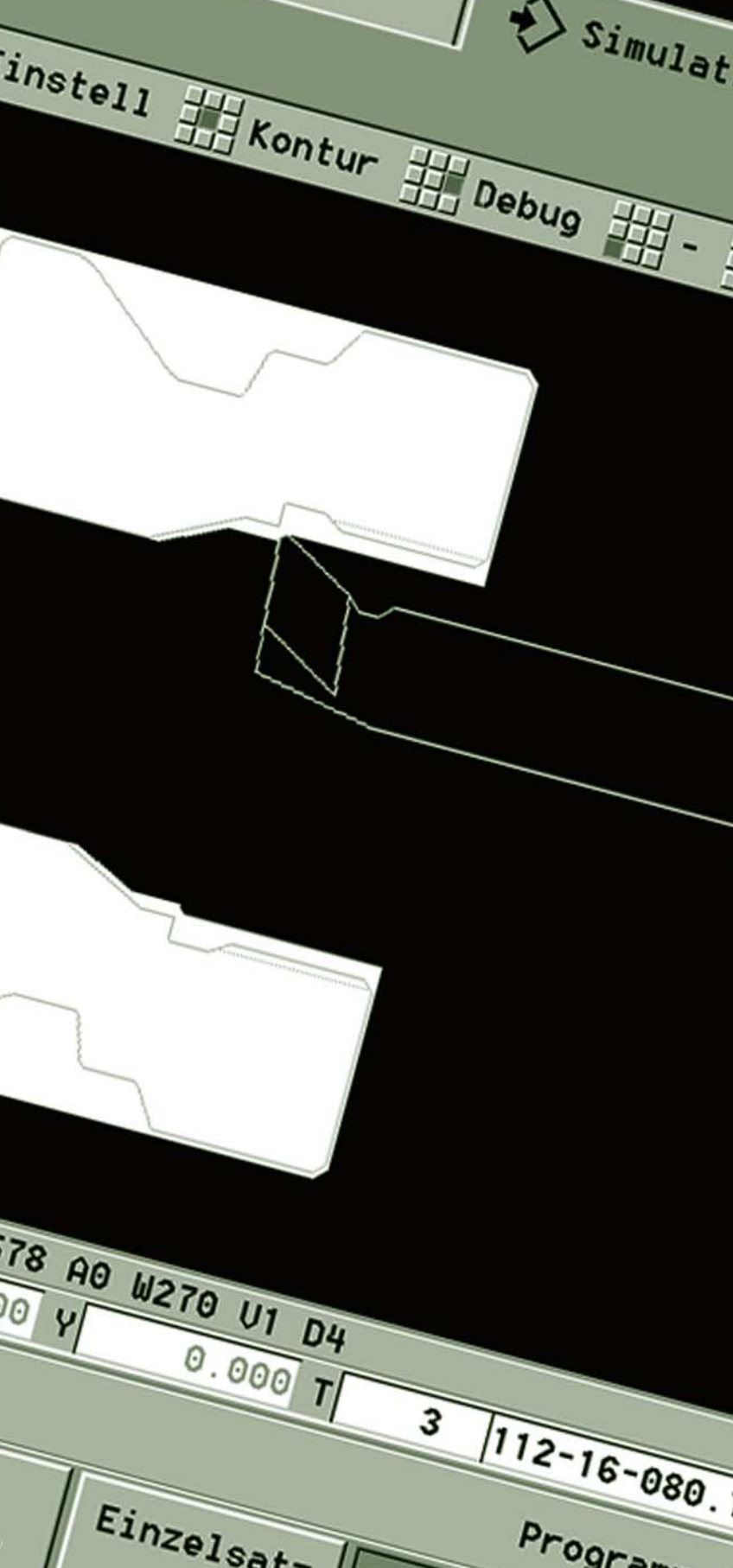
The workpiece is first machined on the front face, then it is rechucked manually. The rear face is machined subsequently.

The expert program mirrors and shifts the contour for the 2nd setup.



PROGRAMMKOPF [PROGRAM HEAD]	Example: Full-surface machining on single-spindle lathe
#SCHLITTEN \$1 [SLIDE]	
REVOLVER 1 [TURRET]	
T1 ID"512-600.10"	
T2 ID"111-80-080.1"	
T4 ID"121-55-040.1"	
SPANNMITTEL 1 [CHUCKING EQUIPMENT] [Zero-point shift Z233]	
H1 ID"3BACK"	
H2 ID"KBA250-86" X100 Q4.	
ROHTEIL [BLANK]	
N1 G20 X100 Z100 K1	
FERTIGTEIL [FINISHED PART]	
. . .	
STIRN Z0 [FRONT]	
. . .	
RUECKSEITE Z-98 [REAR SIDE]	
N20 G308 P-1	
N21 G100 XK5 YK-10	
N22 G101 YK15	
N23 G101 XK-5	
N24 G103 XK-8 YK3.8038 R6 I-5 B0	
N25 G101 XK-12 YK-10	
N26 G309	

BEARBEITUNG [MACHINING]	
N27 G59 Z233	Zero-point shift 1st setup
N28 G65 H1 X0 Z-135 D1	Display chucking equipment of 1st setup
N29 G65 H2 X100 Z-99 D1 Q4	
. . .	
N82 M15	Prepare the rechucking
N83 G65 H1 D1	Delete chucking equipment of 1st set-up
N84 G65 H2 D1	
N86 L"UMHAND" V1 LF98 LH99	Expert program for manual rechucking
	V=
	LF=length of finished part
	LH=distance between chuck reference point
	and edge of workpiece
N88 G65 H1 X0 Z-99 D1	Activate chucking equipment for rear-face machining
N89 G65 H2 X88 Z-63 D1 Q4	
. . .	
N125 M5	Milling - rear side
N126 T1	
N127 G197 S1485 G193 F0.05 M103	
N128 M14	
N130 M107	
N131 G0 X22.3607 Z3	
N132 G110 C-116.565	
N133 G153	
N134 G147 I2 K2	
N135 G840 Q0 NS22 NE25 I0.5 R0 P1	
N136 G0 X154 Z-95	
N137 G0 X154 Z3	
N138 G14 Q0	
N139 M105	
N141 M109	
N142 M15	
N143 M30	
ENDE [END]	



5

Graphic Simulation

5.1 Simulation Mode of Operation

Simulation screen

- 1 Info line: Submode of simulation, simulated NC program
- 2 Simulation window: The machining is depicted in up to three windows
- 3 Programmed NC block (NC source block) – alternative display of variables
- 4 Displays: NC block number, position values, tool information – alternative cutting values
- 5 Coordinate systems of the slides
- 6 Status of the simulation, status of the zero point shift

Functions of the simulation

The Simulation mode shows a graphic representation of programmed contours, the paths of traverse and cutting operations. The CNC PILOT shows the working space, tools and chucking equipment true to scale.

Check machining operations with the C or Y axis in the supplementary windows (front/surface and side view windows).

For complex NC programs with branches, variable calculations, external events, etc., you simulate all inputs and events to test all program branches.

During simulation, the CNC PILOT calculates the **machining and idle-machine times** for every tool.

For lathes with several slides, the **Synchronous point analysis** enables you to optimize your NC program.

Up to four workpieces in the working space

The CNC PILOT supports the program text for lathes with more than one slide in one working space. You can simulate the simultaneous machining of up to four workpieces.

The simulation mode is grouped into:

- **Contour simulation:** Simulation of programmed contours
- **Machining simulation:** Inspection of the machining process
- **Motion simulation:** Simulation of real-time machining with continuous contour regeneration



Soft keys



Change to the DIN PLUS mode



Change to the TURN PLUS mode



Switch to the next slide



Activate the magnify function



Set single-block mode



Set basic block mode



Call next selection



The control parameter 1 ("settings") defines whether the display values will be in millimeters or inches. The setting in "program head" has no influence on operation and display in the simulation mode of operation.

5.1.1 Graphic Elements, Displays

Graphic elements:

■ Coordinate systems

The zero point of the coordinate system corresponds to the workpiece zero point. The arrows of the X and Z axes point in the positive direction. If the NC program is machining more than one workpieces, the coordinate systems of all required slides are displayed.

■ Displayed blank form

- Programmed: Programmed blank form
- Not programmed: "standard blank form" (control parameter 23)

■ Display of finished part (and help contours)

- Programmed: Programmed finished part
- Not programmed: No display

■ Display of tool

- Programmed in the NC program: The tool programmed in the REVOLVER (TURRET) section is used
- Not programmed in the NC program: The entry in the tool list is used (see "3.3 Tool List, Tool Life Data")

The CNC PILOT generates the displayed tool from the parameters of the tool database. Whether the complete tool or only the "cutting area" is shown is specified in "Picture number" (picture no.=−1 in the tool editor: No tool displayed).

■ Chucking equipment display

The simulation graphics displays the chucking equipment provided that it has been programmed with G65.

The CNC PILOT uses the parameters of the chucking equipment database to display a chuck.

■ White dot

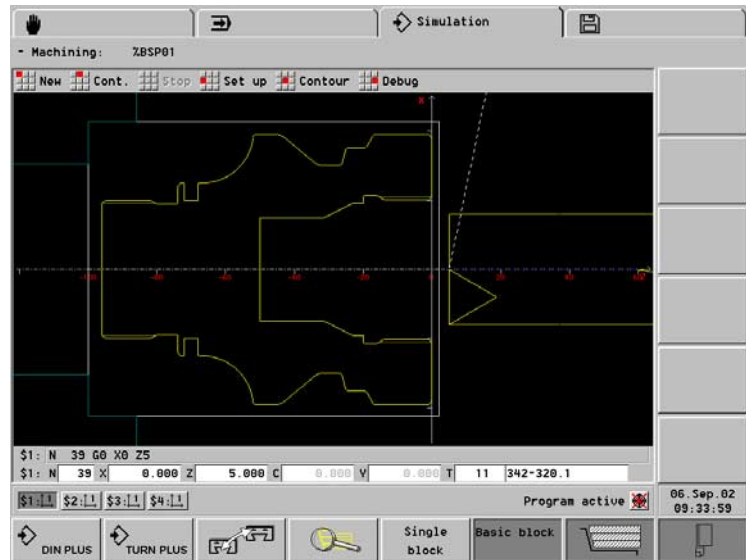
The point of light (small white rectangle) represents the theoretical tool tip.

■ Rapid traverse paths

Are shown as a broken white line.

■ Workpiece line display and tool path display

Tool paths are shown as a continuous line. They describe the path of the theoretical tool tip. The **wire frame graphics** is particularly convenient if you only need a quick overview of the proportioning of cuts. The path of the theoretical tool tip, however, is not identical with the contour of the workpiece. This graphic is therefore not as suitable if you wish to run a thorough check on the machined contour. In the CNC, this "falsification" is compensated by the cutting radius compensation.



Continued ►

In control parameter 24, you can define the color of the feed path for the respective T number.

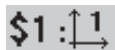
In the **cutting path display**, the CNC PILOT shades the area traversed by the cutting edge of the tool. The cutting path graphic accounts for the exact geometry of the tool tip (cutting radius, cutting width, tool-tip position, etc.).

When using the **cutting path graphics**, you can check whether the contour is machined completely or needs to be reworked, whether the contour is damaged by the tool or overlaps are too large. The cutting path graphics is especially useful for recessing or drilling operations as well as for machining slopes where the tool shape has an essential influence on the accuracy of the resulting workpiece.

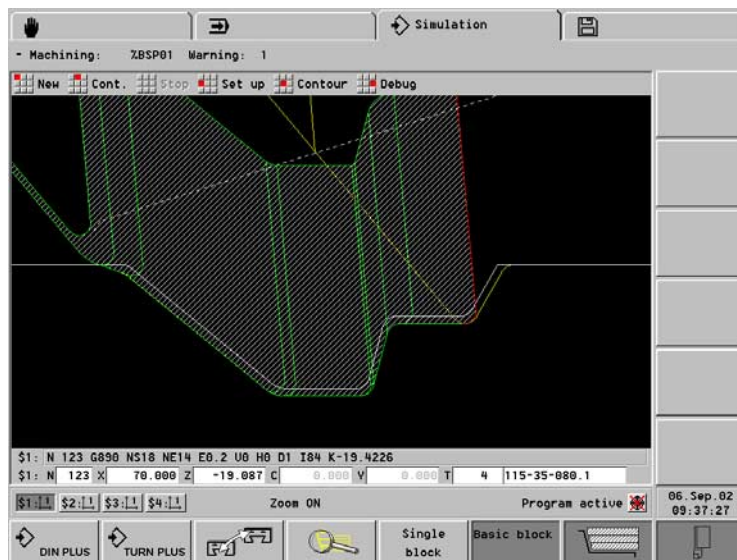
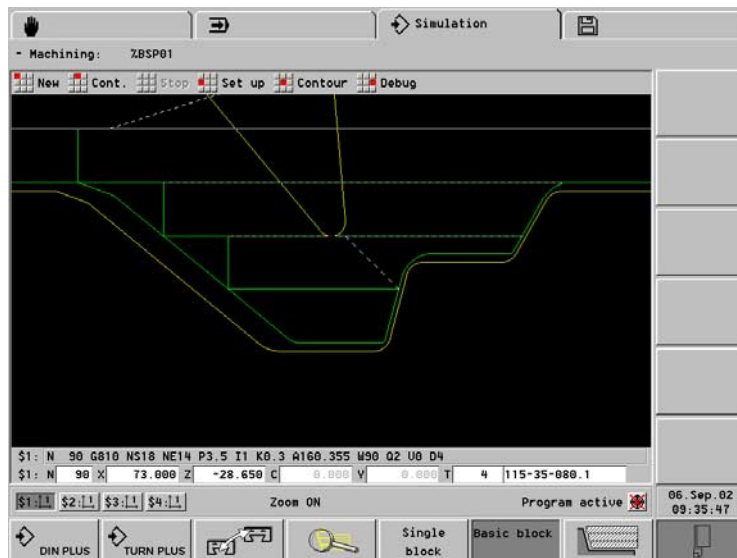
Notes on the display modes

- Programmed NC block (NC source block)
 - Display of NC source blocks of up to four slides (setting: "Set up -Window")
 - Alternatively: Display of four selected variables (selection: "Debug - Display variables - Set variables")
- Displayed information:
 - Block number, position values (actual values) and tool of the selected slide
 - Alternative to the tool data: Spindle speed, feed rate, direction of spindle rotation

Coordinate systems of the slides



- \$n (n: 1..6): Slide code – the selected slide is marked
- Symbol: Configured coordinate system of this slide
- Number in the symbol: Contour, that is machined by this slide



Zero point shift

In the "Contour selection" dialog box ("Set up – Contour selection") you define whether zero point shifts will be accounted for in the simulation. – As an alternative you can use the touch pad to click the "zero point shifts" symbol in order to change the setting.

Changed settings do not become effective until the simulation is restarted.



Including zero point shifts:

- The **machine zero point** is the reference point for the positioning of contours and for the traverse paths.
- Zero point shifts are included in calculation



Zero point shifts **are not included in calculation**:

- The **workpiece zero point** is the reference point for the paths of traverse
- Zero point shifts are ignored

If you use the program section code CONTOUR and G99, no matter what the status of the zero point shift:

- The workpiece (the contour) is depicted at the position defined in CONTOUR
- G99 X.. Z.. shifts the workpiece to a new position

Several workpieces in the working space

The CNC PILOT depicts up to four workpieces in the working space and simulates machining of these workpieces. You define the (first) position of the workpiece in CONTOUR. It is possible to shift the workpiece position later with G99.

Status of the zero point shifts



Zero point shifts are included in calculation



Zero point shifts are not included in calculation



A change of status is not accounted for until the simulation is restarted. The symbols are shown faded as long as the changed setting is not yet in effect.

Coordinate systems of the contours



- Qn (n: 1..4): Contour n – the selected contour is marked
- Symbol: Coordinate system of this contour

5.1.2 Basics of Operation

How to activate the simulation function

- ▶ Load the desired NC program.
- ▶ Set the simulation window (face, surface window, etc.)
- ▶ Set the simulations mode (single block, basic block or without stop)
- ▶ Select the simulation mode (contour, machining, motion).
- ▶ Select "New."
- ▶ Simulation mode "without stop":
 - "Stop" stops the the simulation
 - "Continue" resumes the simulation
- ▶ Simulation mode "Single block or basic block":
 - The simulation stops after every single block/basic block
 - "Continue" resumes the simulation

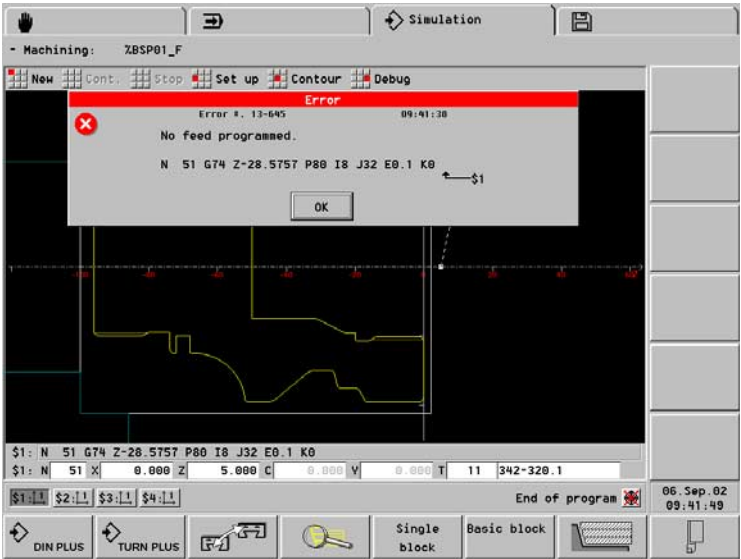
During a **simulation stop** you can switch the block mode, edit the settings or call the dimensioning function.

Errors andWarnings

Warnings that occur during the interpretation of an NC program are displayed in the header. When the simulation has been stopped or completed, you can call up the messages by selecting "Set up - Warnings." If more than one warning has occurred, press ENTER to call up the next message.

The CNC PILOT deletes a warning after you have confirmed the corresponding message with ENTER. The system stores a maximum of 20 warnings.

If an error occurs during the interpretation of an NC program, the simulation is stopped.



"Set simulation modes" soft keys

Single block	Stop after every NC source block. The continue soft key simulates the next NC source block.
Basic block	<ul style="list-style-type: none">■ Contour simulation: Stop after every contour element. Contour macros (contour cycles) are "segmentalized." Select "Continue" to display the next contour element.■ Machining or motion simulation: Stop after each path of traverse. Machining cycles are "segmentalized." The continue soft key simulates the next path of traverse.

Without stop (single block and basic block soft keys are not pressed): The simulation is conducted without stop.

5.2 Main Menu

"Prog(ram selection)" drop-down menu:

- **Load**
 - ▶ Select NC program and press OK
- **From DIN PLUS** – takes the NC program selected in DIN PLUS
- **Menu items** for calling the:
 - Contour simulation: **"Contour"**
 - Machining simulation: **"Machining"**
 - Movement simulation: **"Motion"**
 - 3-D depiction: **"3-D view"**

"Set up" drop-down menu:

Settings that you made previously apply in the contour, machining and motion simulation.

■ "Set up – Window" (window selection dialog box)

Select the combination of windows best suited to the machining that you wish to inspect:

Front window

The contour and traverse paths are shown in the XY plane, taking the spindle position into account. The spindle position 0° is located on the positive X-axis (designation: "XK").

Surface window

The contour display and traverse-path display are oriented to the position on the "unrolled" lateral surface (designation: CY) and the Z coordinates.

Contours with the C-axis are displayed the same way as contours on the workpiece surface. (In the graphic simulation window of the DIN PLUS editor, the surface contours are drawn "on the milling floor" and are therefore shorter than the arc on the workpiece surface.)

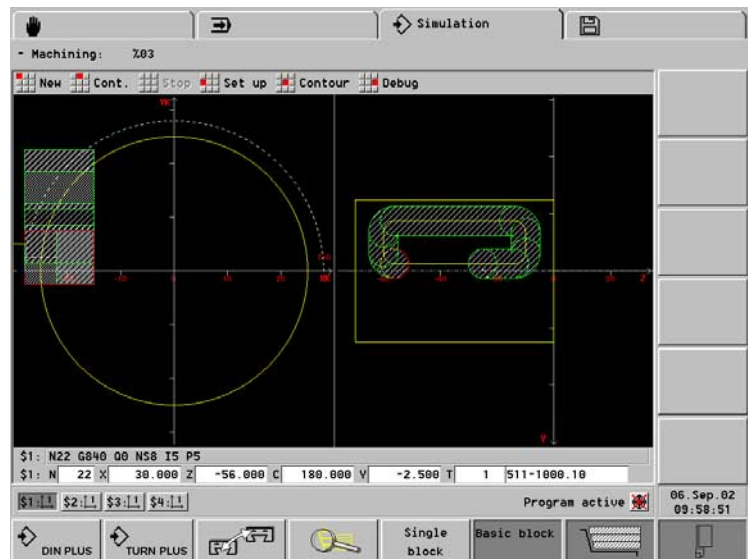
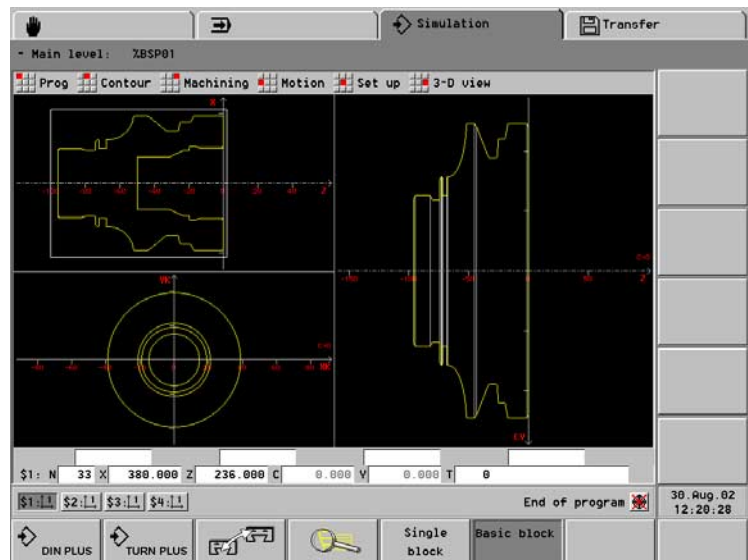
"Side view (YZ)" window

The contour and traverse path are shown in the YZ plane. The side view depicts only the Y and Z coordinates – **not the spindle position**.

Path display in supplementary windows

The front window, surface window, and side view are considered supplementary windows. Traverse paths are not shown until the C axis has been oriented or a G17 or G19 (with Y axis) has been executed.

G18 or a C axis out of orientation **stops** output of the traverse paths in the supplementary windows.



■ After program changes in the DIN PLUS editor, you need only press "New" to simulate the changed NC program.

■ **The front and the surface windows** operate with a "fixed" spindle position. Whereas the machine turns the workpiece, the graphic simulation moves the tool.

■ The "surface window" and the "side view (YZ)" are shown alternatively.

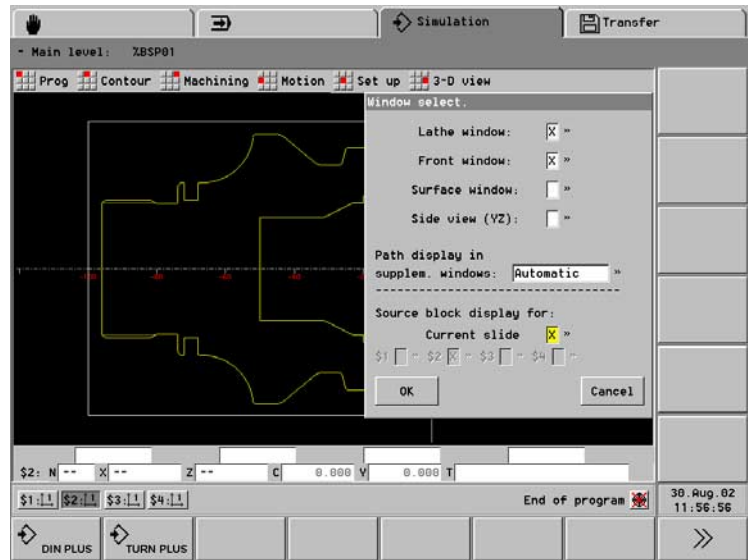
Continued ▶

As an alternative, you can set the "path display in supplementary windows" to **always** (in the "Window selection" dialog box). Then, each traverse path will be shown in all simulation windows.

Source block display

For NC programs with more than one slide, define which slides are to be shown in the source block display.

- **"Set up - Slides"**: For lathes with more than one slides, define:
 - "Path output for ...":
 - "All slides": Display traverse paths of all slides
 - "Current slides": Display traverse paths of the selected slide
 - Slide position: Define for every slide whether the tool traverse should be depicted in front of or behind the workpiece.
 - "Reset" button: The slide position defined in the machine parameters is adopted.
- **"Set up - Contour selection"**:
 - In the dialog box, define whether **one** selected contour or **all** contours of the NC program are described.
 - Define whether zero point shifts are to be taken into account.
- **"Set up - Status line"** or "PgUp/PgDn" switches the die "display." Thus you can check either the tool data or the technology data.
- **"Set up - Zero point C"** (only during active "surface window"): In the "zero point" dialog box, define the position at which the unrolled cylinder is to be "cut open." The "C-angle" that you enter is located on the Z axis.
Default setting: "C-angle = 0°"



5.3 Contour Simulation

5.3.1 Contour-Simulation Functions

The contour-simulation function allows you to:

- Select between "section or view" graphics.
- Check the contour programming through simulation in single blocks.
- Check the parameters of a contour element (element dimensioning).
- Measure each contour point with respect to a reference point (point dimensioning).

Contour simulation presupposes that the blank or finished part contour (blank or finished part definition, auxiliary contours) is programmed. If the blank or finished part contours have not yet been completely programmed, they are displayed as completely as possible.

Esc Back to main menu

■ Menu points or control of the simulation

- **New:** Redraws the contour (program changes are taken into account)
- **Continue:** Displays the next NC source block or basic block

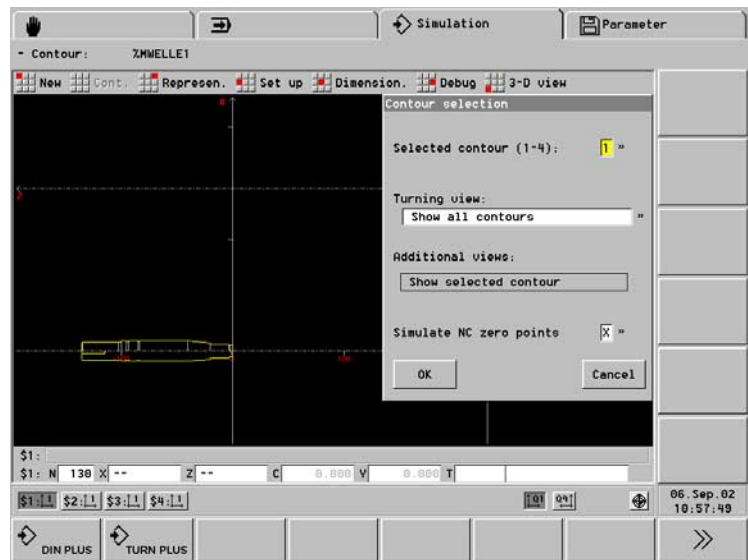
■ Menu item "Representation" (of contour)

You configure:

- (cross) "Section"
- (lateral) "View"
- "Section and view": Above the center of rotation the lateral view, below the center, the cross section

"Set up – ..." drop-down menu:

- **"... – Window":**
"... – zero point C":
 see "5.2 Main Menu"
- **"... – Contour selection":**
 - In the dialog box, define whether **one** selected contour or **all** contours of the NC program will be shown.
 - Define whether zero point shifts will be included.
- **"... – Warnings":** see "5.1.2 Notes for Operation"
- **Menu item "3-D view":** see "5.7 3-D View"
- **"Debug" drop-down menu:**
 If you use variables to describe the contour, you can show and edit them with the debug functions (see "5.8 Checking the NC Program Run").



In single/basic block mode, "section" graphics is active.

5.3.2 Dimensioning

Selection: "Dimensioning" menu item

Esc Back to contour simulation

■ **"Element dimensioning" menu item**

The "Display" line shows all the data of the marked contour element.

■ The arrow indicates the direction of the contour description

■ Move to the next contour element: Arrow left/right

■ Switch contours (example: switch between workpiece blank and finished part contour): Arrow up/down

■ **"Point dimensioning" menu item**

The CNC PILOT shows the dimensions of the contour point with respect to the "reference point."

Set reference point:

► Position the cursor onto the reference point (small red square)

► Select "Set point of reference" – the small square changes its color.

► Position the cursor to the contour point to be measured – the CNC PILOT shows the dimensions with respect to the reference point

Cancel reference point:

► "Point of reference OFF" deactivates the selected reference point. You can now set a new point of reference.

The following rules apply:

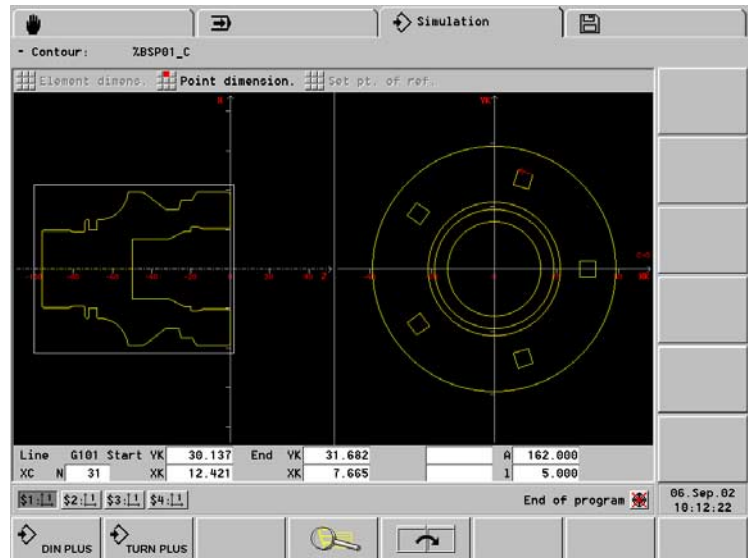
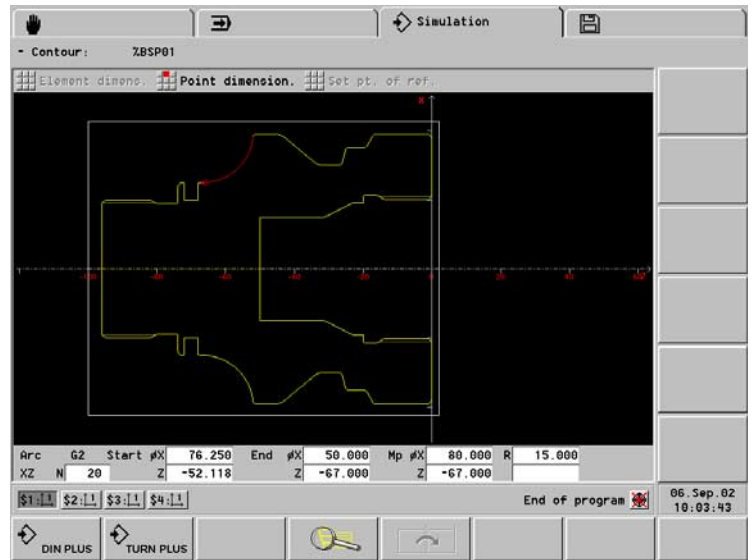
■ Use the vertical arrow keys to move to the next element group.

■ In the case of figures, the individual elements are measured.

■ The selected reference plane (XC, XY, etc.) is shown in the status line.



You can also call the **dimensioning functions** in the Machining/Motion mode of simulation ("Dimensioning" menu item).



Special soft keys



Change to next simulation window. Prerequisite: There are contours on the reference planes (front face, Y face, lateral surface, side view).

5.4 Machining Simulation

Functions of the machining simulation:

- Checking the tool traverse paths
- Checking the proportioning of cuts
- Ascertaining the machining time
- Monitoring for violations of the protection zones and traverse limits
- Finding and setting variables
- Saving machined contours

Esc Back to main menu

Protection zone and limit switch monitoring

In addition to the setting in the simulation, the protection zone monitoring is also switched on in the machine parameter 205, ... ("Monitoring on/off"). You set the protection zone dimensions in the Setup mode (Manual Control operating mode). The dimensions are managed in the machine parameters 1116, ...

Contour generation during simulation

You can save contours generated in the simulation and transfer them into the NC program. Example: You describe the blank form and finished part and simulate machining of the first setup. Then you save the contour. You define a shift of the workpiece zero point and/or a mirror image. The simulation save the "generated contour" as the workpiece blank and the originally defined finished part contour - taking the zero point shift and mirroring into account.

In DIN PLUS, you insert into the program the workpiece blank and finished part contour that you generated during simulation (block menu "Insert contour").

Menu items for controlling the simulation

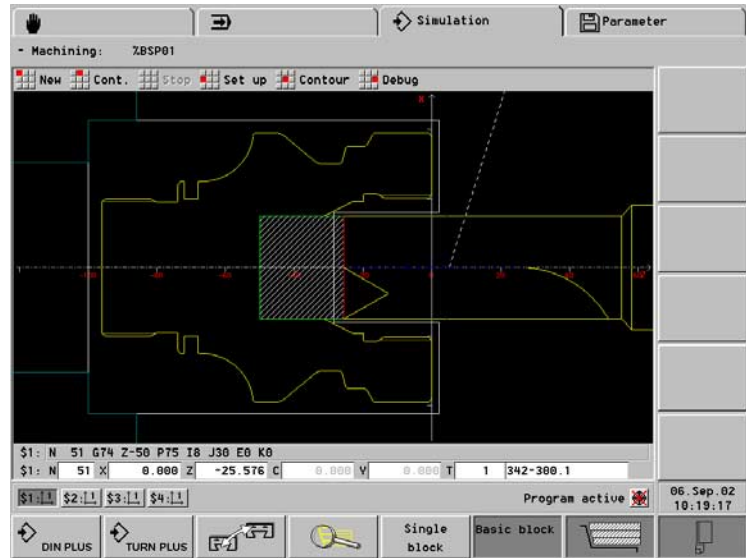
- **New:** Start a new simulation (program changes are taken into account)
- **Continue:** Simulates the next NC source block or basic block
- **Stop:** Stops the simulation. Thus you can edit the settings or use the "contour follow-up" function.

"Set up – ..." drop-down menu

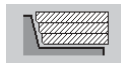
- **"... – Window":**
- **"... – Slides":**
- **"... – Contour selection":**
- **"... – Status line":**
- **"... – Zero point C":**
See "5.2 Main menu"
- **"... – Warnings":** see "5.1.2 Basics of Operation"



You can change the speed of the machining simulation with control parameter 27.



Special soft keys



View of the paths of traverse: **Line** or **(cutting) track**



Tool depiction: **Point of light** or **tool**

Continued ►

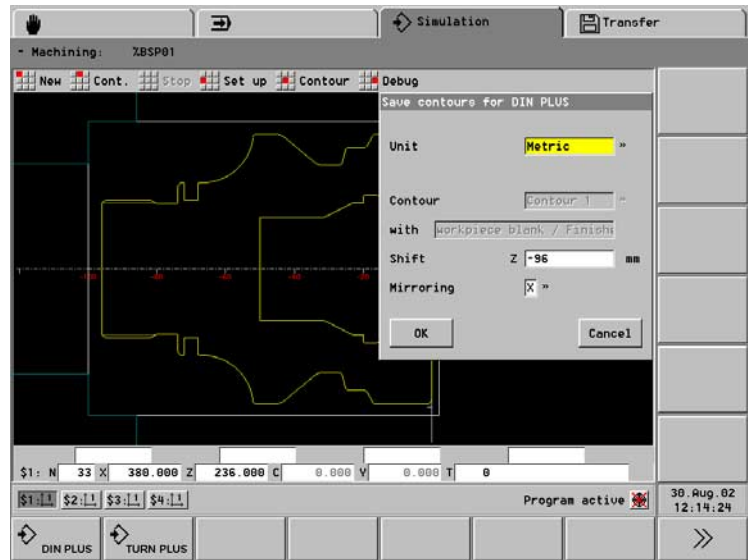
- **"... – Times"**: see "5.9 Time Calculation"
- **"... – Protection zone – ..."**
 - **"Monitoring off"**: Protection zones/software limit switches are not monitored
 - **"Monitoring with warning"**: The CNC PILOT registers protection zone violations or limit switch violations and handles them as warnings. The NC program is simulated up to the end of program.
 - **"Monitoring with error (message)"**: A protection zone or limit switch violation results in an immediate error message and cancelation of the simulation.

"Contour – ..." drop-down menu:

- **"... – Contour follow-up"**
Updates the contour to depict the current progress of machining. The CNC PILOT takes the blank part as a basis and accounts for each cut.
- **"... – Dimension."**: see "5.3.2 Dimensioning"
- **Menu item "3-D view"**: see "5.7 3-D View"
- **"... – Save contours"**
Saves the contour in the simulated machining condition as BLANK and the programmed finished part.
Settings in the "Save contours for NC program" dialog box:
 - Unit: Contour description in millimeters or inches
 - Contour: Selection of the contour (is more than one contour exists)
 - Shift: Value of the workpiece zero point shift
 - Mirroring: Contours are mirrored/not mirrored

"Debug" drop-down menu

If you use variables to machine the workpiece, you can show and edit them with the debug functions (see "5.8 Checking the NC Program Run").



5.5 Motion Simulation

The motion simulation depicts the workpiece blank material as a "filled surface" and "machines" it during simulation by "erasing" the material. The tools move at the programmed feed rate (program-run graphics).

You can interrupt the motion simulation at any time, even during simulation of an NC block. The display below the simulation window indicates the target position of the current path.


If other simulation windows are activated in addition to the lathe window, the machining process is shown as "track-display graphics" in the supplementary windows.

Protection zone and limit switch monitoring

In addition to the setting in the simulation, the protection zone monitoring is also switched on in the machine parameter 205, ... ("Monitoring on/off"). You set the protection zone dimensions in the Setup mode (Manual Control operating mode). The dimensions are managed in the machine parameters 1116, ...

Visual limit switch monitoring

Depending on the "limit switch for slide x" setting ("slide settings" dialog box) the motion simulation shows the positions of the **software limit switches** relative to the tool point (red rectangle). That simplifies monitoring the traverse paths near the working space limits. The visual limit-switch monitoring is independent from the protection zone monitoring and limit switch monitoring.ng.

 Back to main menu

Menu items for controlling the simulation

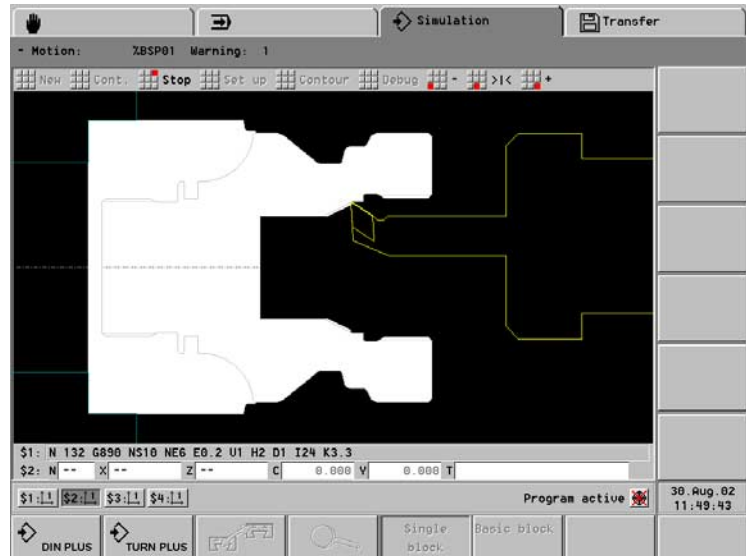
- **New:** Start a new simulation (program changes are taken into account)
- **Continue:** Simulates the next NC source block or basic block
- **Stop:** Stops the simulation. Thus you can edit the settings or use the "contour follow-up" function.

"Set up – ..." drop-down menu:

- **"... – Window":**
 - **"... – Contour selection":**
 - **"... – Status line":**
see "5.2 Main Menu"
 - **"... – Slide":** See "5.2 Main Menu."
- In the motion simulation you can also activate the "limit switch display for slide x."



The simulation shows the limit switch dimensions relative to the tool point. This is why the limit switch dimensions are repositioned in a tool change.



- **"... – Warnings":** see "5.1.2 Basics of Operation"
- **"... – Times":** see "5.9 Time Calculation"
- **"... – Protection zone – ...":** see "5.4 Machining Simulation."

"Debug" drop-down menu

If you use variables to machine the workpiece, you can show and edit them with the debug functions (see "5.8 Checking the NC Program Run").

"Contour" drop-down menu:

- **"Contour - Dimension":** see "5.3.2 Dimensioning"
- **"Contour – 3-D view":** see "5.7 3-D View."

Influence the traversing speed (by menu)

- **"-":** slows the traversing speed.
- **">|<":** means traversing speed in real time.
- **"+":** increases the traversing speed.

5.6 Zoom Function

The zoom function can be used to magnify/reduce the displayed graphic or isolate a detail.

Zoom settings by keyboard

Prerequisite: Simulation in "stop condition"



When you call the zoom function, a red frame appears with which you can select the detail you wish to isolate.



With more than one simulation window:
Set the windows

Detail:

- Enlarge: "Page forward"
- Reduce: "Page back"
- Shift: Cursor keys

Zoom settings by touch pad

Prerequisite: Simulation in "stop condition"

- Position the cursor to one corner of the section
- While holding the left mouse key, pull the cursor to the opposite corner of the section

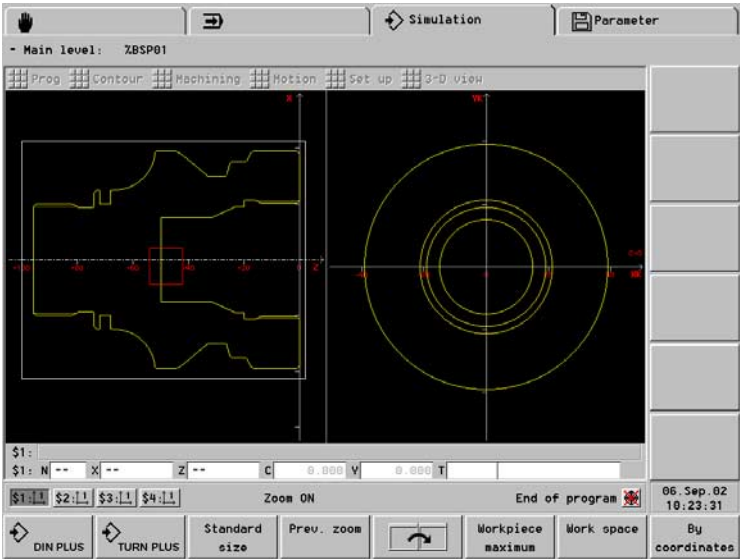
Right mouse key: Return to standard size

Standard settings: See soft-key table



Exit the zoom function

After having enlarged a detail to a great extent, select "Workpiece maximum" or "Work space," and then isolate a new detail.



Soft keys

Standard size	Cancels the zoom settings and displays the last setting ("Workpiece maximum" or "Work space").
Prev. zoom	Cancels the last magnification/setting. You can select "Previous zoom" more than once.
	Switches the zoom function to the next simulation window.
Workpiece maximum	Shows the workpiece in the largest possible view.
Work space	Shows the working space including the tool change position.
By coordinates	Set the "dimensions" of the simulation window and position of the workpiece zero point. If there is more than one window, each window must be set separately. The setting applies to the contour of the selected slide.

5.7 3-D View

In the 3-D view the CNC PILOT shows the workpiece in its simulated condition. If you call the 3-D view from the contour simulation, the finished part is depicted.

Call: "3-D view" menu item

Esc Exit the 3-D view

Switching the display modes:

- With a soft key you can switch between a standard solid model or a "grating" wire model
- To magnify: Soft key or "page up"
- To reduce: Soft key or "page down"
- To rotate: cursor keys, plus and minus key
- Soft key "standard 3-D view" displays the workpiece in the standard size and position

ar




The 3-D view accounts for contours machined by turning – no C or Y operations.

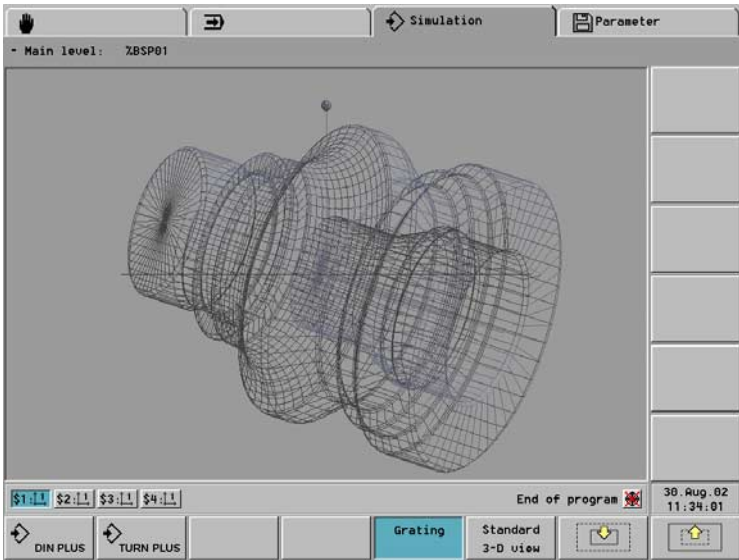
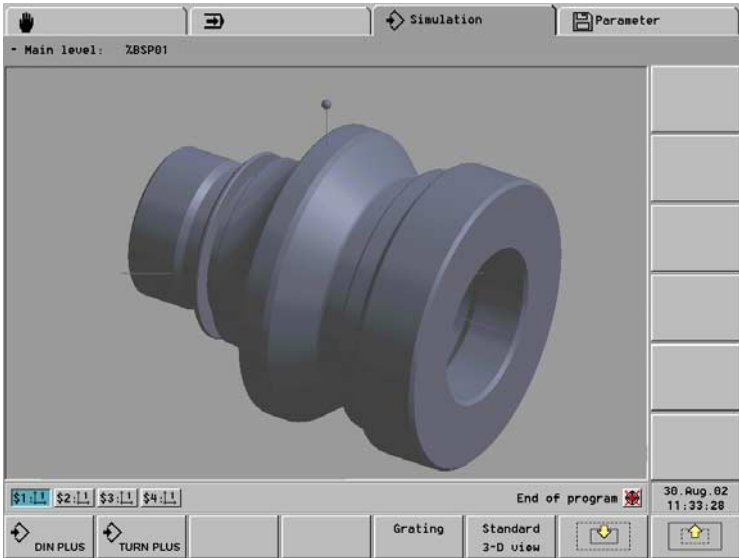
Soft keys

Standard 3-D view Depiction as "solid-body model" in the standard view (not rotated, not enlarged/reduced)

Grating Depiction as wire model

 Magnify displayed graphics

 Reduce displayed graphic



5.8 Checking the NC Program Run

For complex NC programs with branches, variable calculations, events, etc., you simulate all inputs and events to test all program branches.

"Debug" drop-down menu:

- **"Debug – Set start block"**
- **"Debug – Delete start block"**
- **"Debug – View start block"**

If a start block is defined, then up to this block the NC program is compiled and the traverse is not depicted. The CNC PILOT stops – The continue soft key resumes the simulation.

- **"Debug – Variables/Source block"**

In the standard setting, the NC source block is displayed below the simulation window. With "Variables/source block" you can change between the display of four "selected variables" and the NC source block.

- **"Debug – Variables display – ..."**

- **"... – All # variables"**

- ▶ The variables are shown in a dialog box.
- ▶ Use "arrow up/down" and "page up/down" to display the desired variables.

If merely the variable number is displayed, the D variable is not used.

- **"... – All V variables"**

- ▶ Select variable groups and define the "first variable number" (dialog box "V display")
- ▶ The variables are shown in a dialog box.
- ▶ Use "arrow up/down" and "page up/down" to display the desired variables.

- **"... – Set display"**

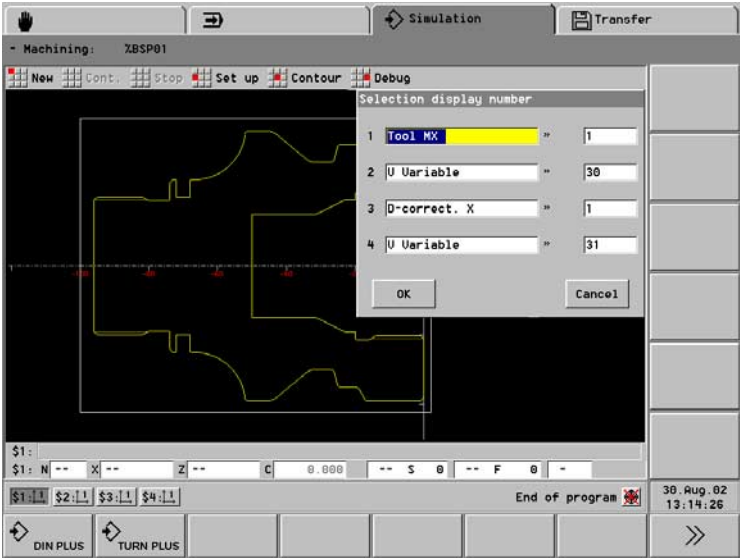
- ▶ Set variable type and number
- ▶ The variables are displayed (alternative to the "NC source block").

- **"... – Reset display"**

Variable fields remain empty.



The variables and events are **simulated**. In other words, the variables used in the Automatic and Manual Control operating modes are not affected.



Variable groups

Selection	Display	Meaning
# Variable	# ..	# variable
V variable	KV ..	V variable
Tool compensation X, ...	KD X, ...	Tool compensation
Machine dimensions X, ...	KM X, ...	Machine dimensions
Tool dimensions Mx, ...	KTM X, ...	Tool dimensions
Sequential events	–	Events of the tool life management and start block search
External events	–	External events

■ "Debug – Change variables – ..."

■ "... Change V variables"

- ▶ Set the variable type and number
- ▶ Preset the "value" or the "event"
- ▶ Define the "status":

– **Undefined:** The variable is not assigned to any value/event. That is the same as the condition after the NC program start. Each time an NC block containing this variable is simulated, the simulation asks you to enter the variable value/event.

– **Defined:** During simulation of an NC block with this variable, the entered value/the event is assumed.

– **Request:** Each time an NC block containing the variable is simulated, the simulation requests the variable value/event..

■ "... – delete all xx variables/events"

If variables have the "status defined," you delete the status of the corresponding variable group/ events.

"xx" stands for:

- V var.: V variables
- D var.: Tool compensation
- E var.: sequential events and external events
- M var.: machine dimensions
- T var.: Tool dimensions

■ "Debug – V variable display"

Provides the variables in the "V variable display" (program head) for editing. By pressing "reset" you load the "preset values."

Prerequisite: The "V variable display" was defined.

■ "Debug – Output window – ..."

■ "... – Activate window"

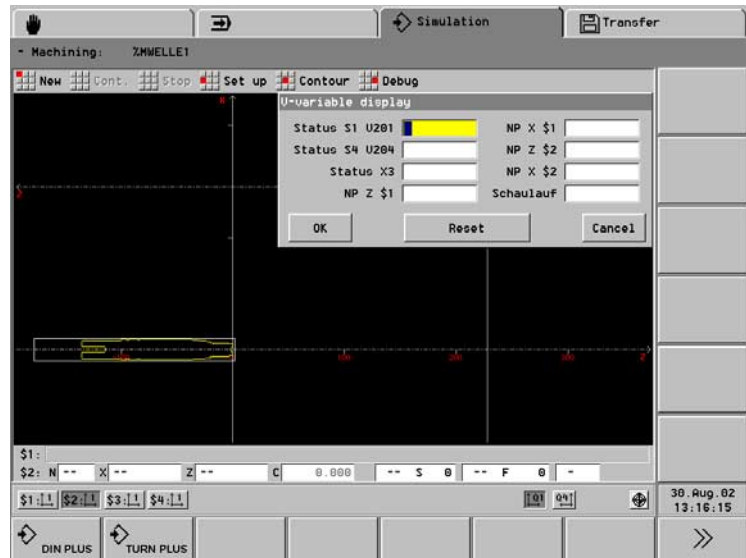
■ "... – Deactivate window"

If the NC program includes data output, you define whether the output window is displayed or suppressed.

■ "... – Display # output"

■ "... – Display V output"


If the data output of # variables and V variables interfere, use these menu items to put the desired display in the foreground..



5.9 Time Calculation


During machining or motion simulation, the CNC PILOT calculates the productive and non-productive times.

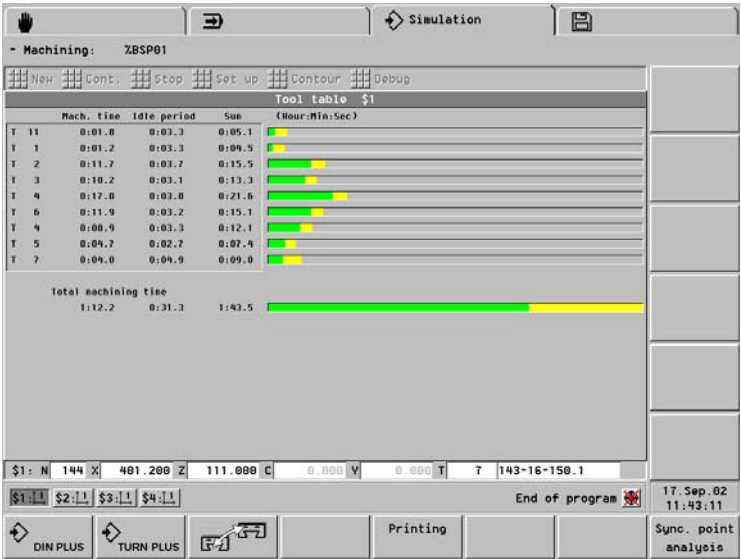
Call: "Settings -Times" menu item

 Exit the time calculation


The machining times, idle times and total times are shown in the table "Time calculation" (green: machining times; yellow: idle times). Each line represents the use of a new tool (for each tool call with T).


If there are more table entries than fit on a screen page, you can call further time data with the arrow keys and PgUp/PgDn.

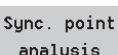
 In control parameters 20 and 21 you can define which control parameters will be included in the time calculation.



Soft keys

 Switch to the next slide

 Output of the time calculation table on a printer (see control parameter 40).

 Call the "synchronous point analysis"

5.10 Synchronous Point Analysis

If you use more than one slide for your machining operation, you can coordinate the machining process with "synchronous points."

The "Synchronous point analysis" analyzes the mutual dependence of the slides. The bar graph displays tool changes, synchronous points and waiting times. Additional synchronous point information describes the selected point (arrow below the bar graphic).

Sync. point analysis

Call: The synchronous point analysis is a subfunction of the time calculation function.

Select synchronous points:

- Change slides: by soft key or upward/downward arrow
- Next/previous synchronous point: arrow left/right

Synchronous point information:

- NC program subprogram
- Active tool
- The NC block relevant for the selected synchronous point
- "tw": Waiting time at this synchronous point
- "tg": Calculated execution time as of program start

Return to time calculation: Press soft key again:

Esc Return to simulation



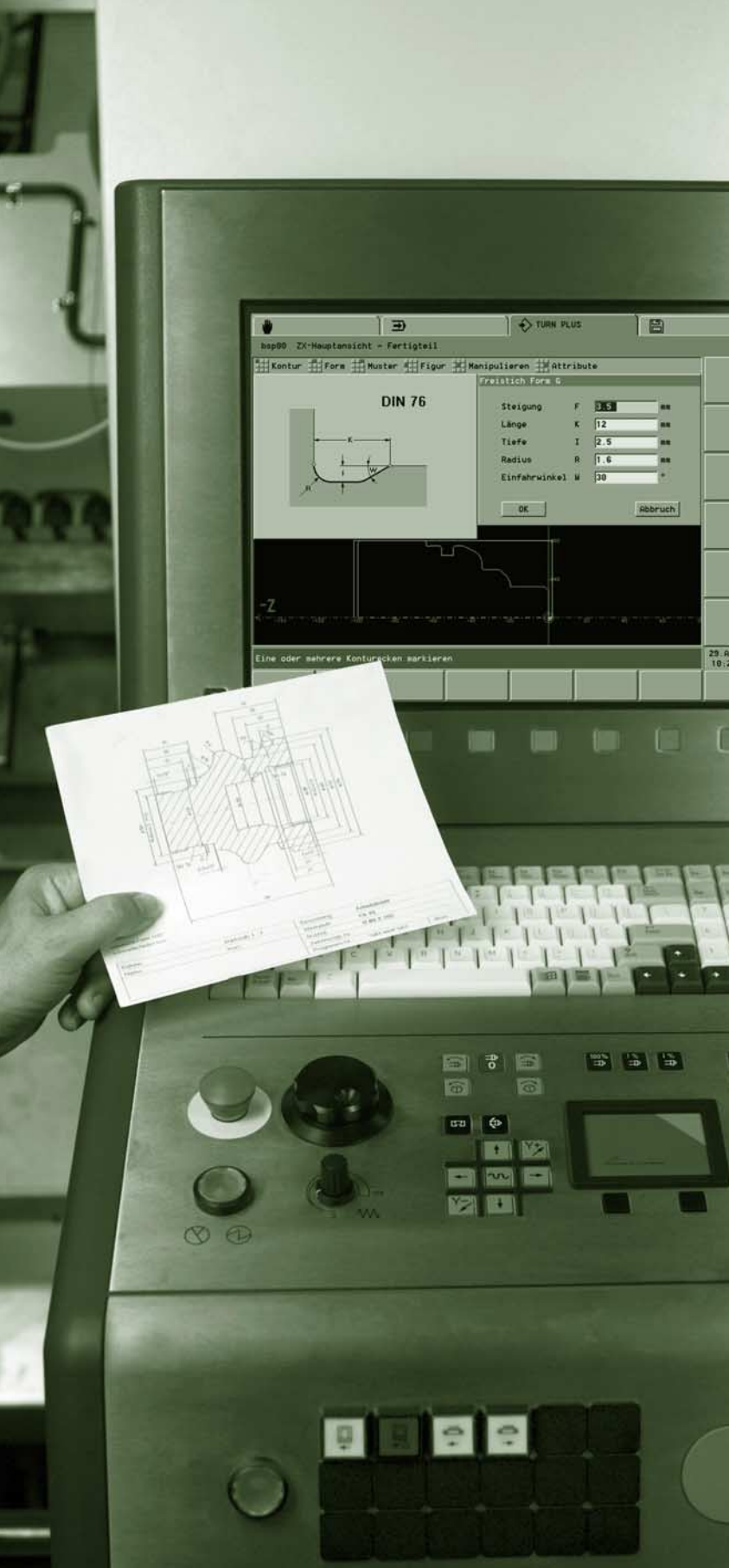
Soft keys



Switch to the next slide

Sync. point analysis

Return to "time calculation"



6

TURN PLUS

6.1 TURN PLUS Mode of Operation

TURN PLUS enables you to describe the workpiece blank and finished part in a graphic-interactive environment. Then you can have the working plan generated fully automatically or generate it yourself interactively. The result is a commented and structured NC program.

TURN PLUS comprises:

- Graphic-interactive contour creation
- Workpiece setup
- Interactive working plan generation (IWG)
- Automatic working plan generation (AWG)

for

- Turning operations
- Milling and drilling operations with the C axis
- Milling and drilling operations with the Y axis
- Full-surface machining

TURN PLUS concept

The working plan generation is based on the workpiece definition (blank/finished part, milling/drilling contours). During chucking of the workpiece, the cutting limits are determined. For tool change, TURN PLUS provides the following strategies:

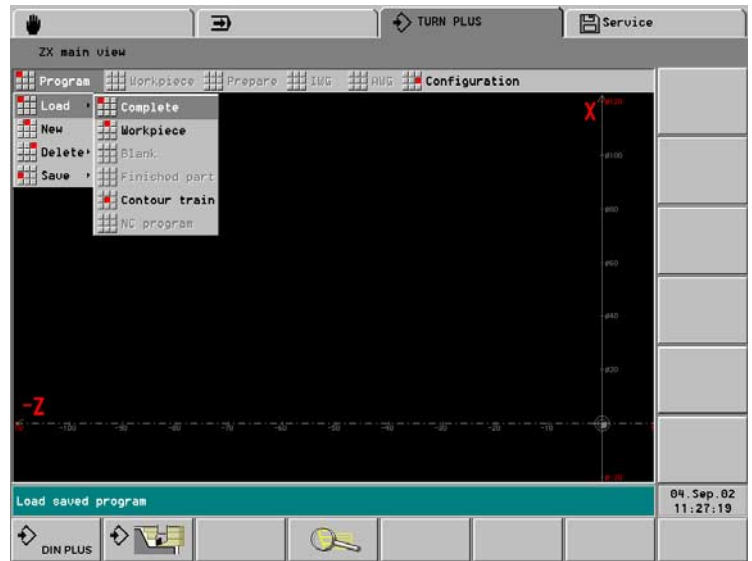
- Automatic selection from the tool database
- Use of the current turret assignment
- TURN PLUS turret assignment

The technology database provides the cutting data.

TURN PLUS generates the working plan, which includes technology attributes such as oversizes, tolerances, peak-to-valley height, etc. into account. Each entry and working step are displayed and can be corrected immediately.

On the basis of **workpiece blank regeneration**, TURN PLUS optimizes the paths for approach and avoids idle cuts or collisions between workpiece and cutting edge. The generation strategy is defined in the "machining sequence" or "machining parameters." This allows you to adapt TURN PLUS to your individual needs.

You can use part of the functions and continue working in DIN PLUS (example: Define a contour using TURN PLUS and program the machining operation in DIN PLUS). Alternately, you can optimize the DIN PLUS program generated by TURN PLUS.



Notes on using TURN PLUS:

The "status line" (above the soft key row) informs you how to proceed.

TURN PLUS uses a multi-level menu structure. The "ESC" key switches one level back.

This description includes operation by mouse, soft keys and touch pad. However, you can still operate the CNC PILOT, as in earlier versions, without soft keys or touch pad.

If **more than one window** (view) is displayed on the screen, the "active window" is identified by a green frame. PgUp/PgDn switches between the windows. The period key "." displays the active window as a full screen. Pressing the period key "." again switches back to multiple windows.

The "Configuration" menu allows you to select the desired type of display and input (see "6.15 Configuration").



The TURN PLUS working plan generation uses the tool/chucking equipment and the technological information contained in the respective database. Therefore, make sure that the database contains an up-to-date and correct description of the operating resources.

6.2 Program Management

6.2.1 TURN PLUS Files

TURN PLUS has indexes for:

- Complete programs (blank and finished part definition as well as working plan generation)
- Workpiece descriptions (workpiece blank and finished part)
- Workpiece blank descriptions
- Finished part descriptions
- Individual contour trains
- TURN PLUS turret assignments (see "6.11.2 Set UpTool List")

You can use this structure for your requirements. for example, for generating different working plans using the same workpiece definition.

"Program (management)" drop-down menu:

■ Load

- ▶ Select the drop-down menu (Complete, Blank, Finished part or Contour train)
- ▶ Select the file

■ New – creates a new TURN PLUS program.

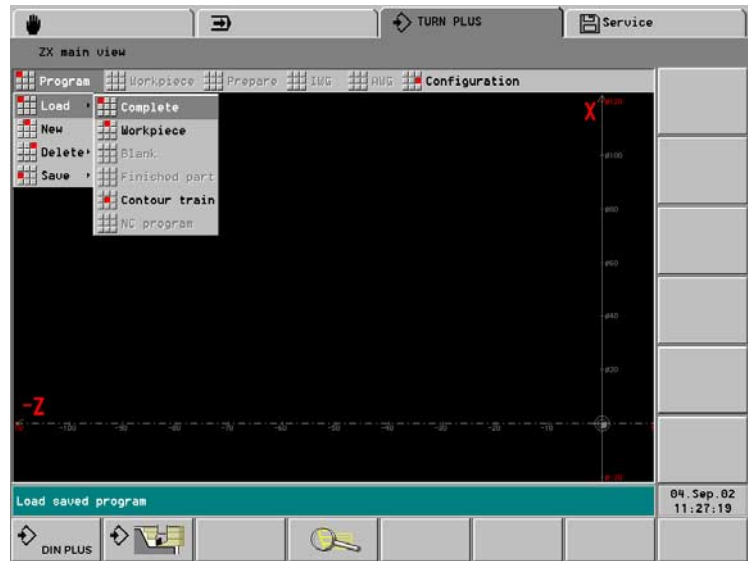
- ▶ Enter the program name and define the material
- ▶ To activate the "Editing program head" window, select "Program head."
- ▶ After completing the "program head editing," define the workpiece blank and finished part and generate the working plan

■ Deleting

- ▶ Select the drop-down menu (Complete, Blank, Finished part or Contour train)
- ▶ Select the file and delete it

■ Saving – saves the created program.

- ▶ Select the drop-down menu (Complete, Workpiece, Blank, Finished part or NC program) - with "Complete" the NC program will also be saved.
- ▶ Enter/check the program name
- ▶ Press "OK" – the file is saved



Soft keys



Change to the DIN PLUS mode



Change to the simulation operating mode



Activate zoom (see: "6.14 Control Graphics")

6.2.2 Program Head

The PROGRAM HEAD comprises:

- **Material** – for determining the cutting values
- **Assignment of spindle to slide for 1st setup**
- **Assignment of spindle to slide for 2nd setup**
For full-surface machining, enter the spindle/slide for machining the setup.
- **Spindle speed limit:**
 - No input: SMAX is the spindle speed limit
 - Input < SMAX: Input is the spindle speed limit
 - Input > SMAX: SMAX is the spindle speed limit**SMAX:** see machining parameter 2 (global technology parameter – spindle speed limit).
- **"M functions" button:** You can define up to five M functions which TURN PLUS will take into account when generating the NC program.
 - At the beginning of the machining process
 - After a tool change (T command)
 - At the end of the machining process

The fields

- Clamping diameter
- Unclamping length
- Clamping pressure

are calculated and automatically entered by TURN PLUS in the "setup" function (see "6.11.1 Clamping the Workpiece").

The other fields contain **organizational information** and **set-up information**, which do not influence the machining process.

Information contained in the program head is preceded by "#" in the DIN program.

The screenshot shows the 'Program head' configuration window in the TURN PLUS software. The window has a title bar with 'ZK main view' and a 'Service' button. Below the title bar is a menu bar with 'Program', 'Workpiece', 'Prepare', 'IUG', 'HUG', and 'Configuration'. The 'Configuration' tab is selected. The main area contains several input fields and buttons. On the left, there are fields for 'Material' (set to 'St 37-k'), 'Machine', 'Drawing', and 'Setup' (with a 'With slide' checkbox). On the right, there are fields for 'Workpiece', 'Company', 'Author', 'Date' (set to '04.09.02'), and two sets of fields for '1st setup' and '2nd setup' (each with 'Spindle', 'With slide', 'Clamping #', 'Free length', and 'Clamp. press.' fields). At the bottom, there are fields for 'Program for automatic lathe' (set to 'No') and 'Speed limit' (set to 'rpm'). There is also a 'Comment' field and an 'M functions' button. At the very bottom, there is a status bar with the text 'Invalid ResourceID' and a date/time stamp '04. Sep. 02 11:30:02'. Navigation buttons like 'DIN PLUS' and 'OK/Cancel' are also visible.

6.3 Workpiece Description

Basics of contour definition

You program a contour by entering individual contour elements one after the other in the correct sequence.

You can define the contour elements/contour position using absolute, incremental, Cartesian or polar coordinates. You can usually program a contour with the dimensions given in the workpiece drawing.

X values are entered as diameter or radius values (see "6.14 Configuration").

TURN PLUS automatically calculates all missing coordinates, points of intersection, center points, etc. that can be derived mathematically. If the entered data permit several mathematically possible solutions, you can inspect the individual solutions and select the proposal that matches the drawing.

Importing contours in DXF format

Contours available in DXF format can be imported into the TURN PLUS programming mode of operation (see "6.8 Importing DXF Contours").

DXF contours describe

- Workpiece blanks
- Finished parts
- Contour train
- Milling contours

6.3.1 Entering the Contour of a Blank Part

- Standard forms (bars, pipes): Definition with workpiece blank macros
- Complex workpiece blanks: Description as for finished part
- Cast or forged blanks are generated from the finished part and the oversize

Input of workpiece blank contour (standard form)

Select "Workpiece - Blank - Bar/Tube."

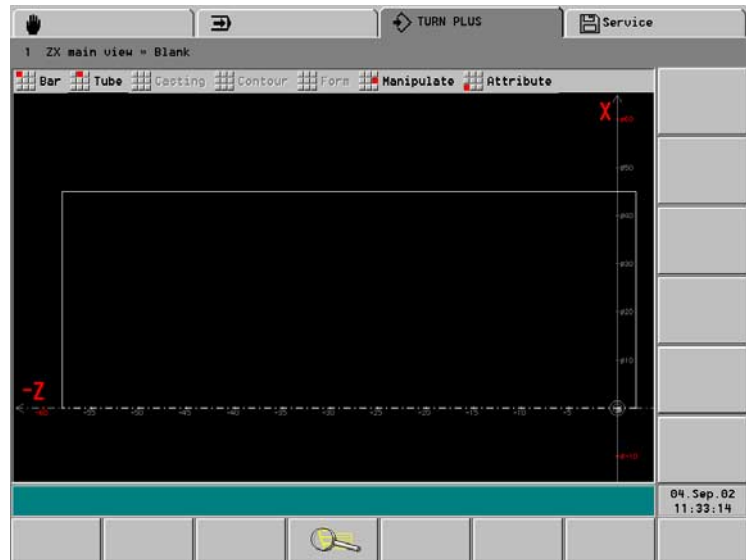
Enter the dimensions of the workpiece blank

The CNC PILOT displays the workpiece blank.

To return to the main menu, press the ESC key.

See also

- "6.4 Contours of Workpiece Blanks"
- "6.9.1 Attributes for Workpiece Blanks"



6.3.2 Input of the Finished Part Contour

The finished part contour includes:

- Turning contour comprising
 - Basic contour
 - Form elements (chamfers, roundings, undercuts, recesses, threads, centric bore holes)
- C-axis contours

Turning contours (blank/finished part) must be **closed**.

Entering the basic contour

Entering the basic contour

Select "Workpiece - Finished part - Contour."

Define the "starting point of contour"

Select "Line/Arc."

Enter the basic contour element for element:

Line

- Use a menu symbol to select a direction
- Describe a line

Arc

- Use a menu symbol to select the direction of rotation
- Describe the arc

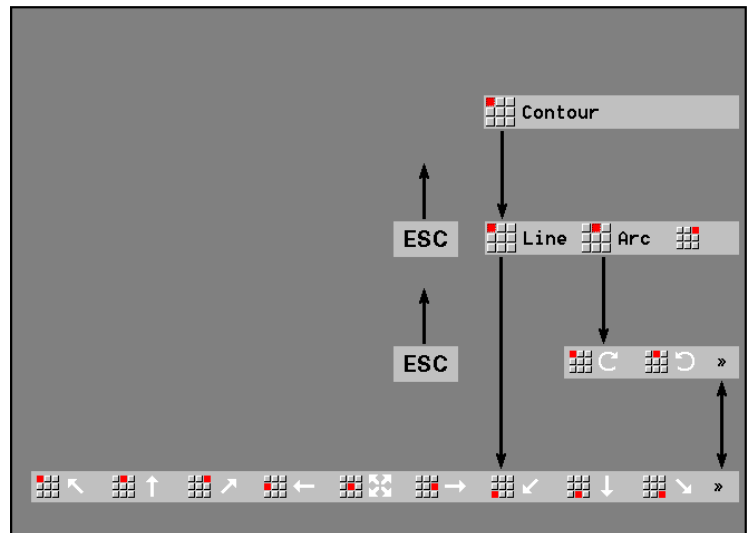
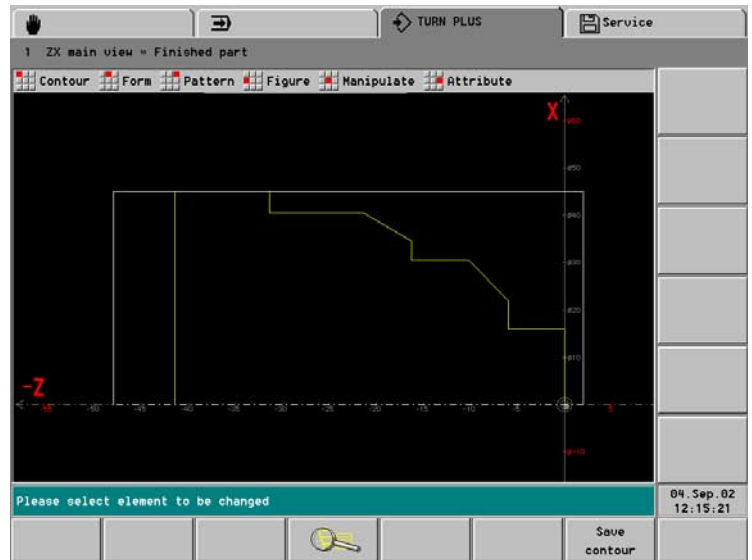
- **Switch** between line/arc menu: by soft key

If the contour is **not closed**:

- Press the ESC key twice
- Answer "yes" to "Close contour?"

See also

- "6.5.1 Basic Contour Elements"
- "6.3.7 Help Functions for Element Definition"
- "6.9 Assigning Attributes"



Describe first the basic contour and then superimpose the form elements.

6.3.3 Superimposing form elements

Form elements are **superimposed** on the basic contour. There remain independent elements that can be edited or deleted. If required, TURN PLUS generates a special machining cycle for the form elements.

The position selection considers the type of form element:

- **Chamfer:** Outside corners
- **Rounding:** Outside and inside corners
- **Undercut:** Inside corners defined by paraxial right-angled lines
- **Recess:** Straight lines
- **Threads:** Straight lines
- **(Centric) bore hole:** Center axis on front and rear face

Superimposing form elements

Select "Workpiece - Finished part - Form."

Select the desired form element from the drop-down menu.

Select a form element

Select a position by soft key or touch pad

Select more than one form element

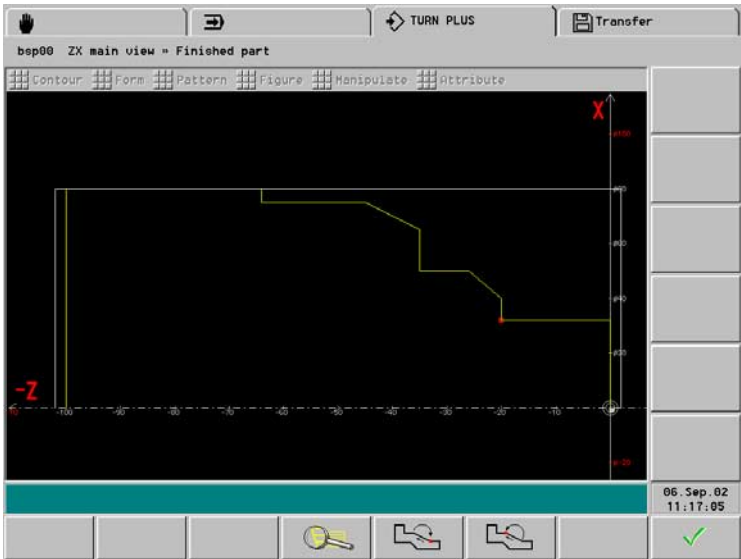
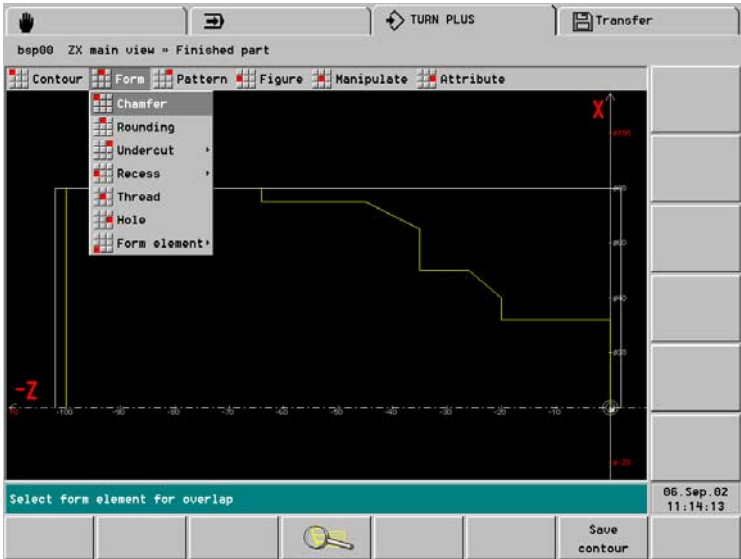
Select positions by soft key or touch pad

Enter the parameters of the form element

TURN PLUS depicts the form elements.

See also

- "6.5.2 Form Elements"
- "6.3.6 Notes on Operation"



Define chamfers, rounding arcs, undercuts, etc. as **form elements**. Then the working plane generation can take into account special operations on these form elements.

6.3.4 Integrating a Contour Train

You can program frequently occurring **contour trains** once and integrate them once as a series (row) in the contour. Integrated contour trains are part of the contour.

The contour trains (overlay elements)

- Circular arc
- Wedge
- Pontoon

are predefined. You can describe complex contours like a finished part contour. To be able to use a contour train in different programs, save the respective contour train.

Overlay elements superimpose existing linear or circular contour elements (supporting contour elements).

Integrating a Contour Train

Loading a contour train (if desired):

Select "Program – Load – Contour train"

Select and load file

Return to main menu

Select "Workpiece – Finished part – Form – Form element – ..."

Standard overlay element:

Select and describe the overlay contour

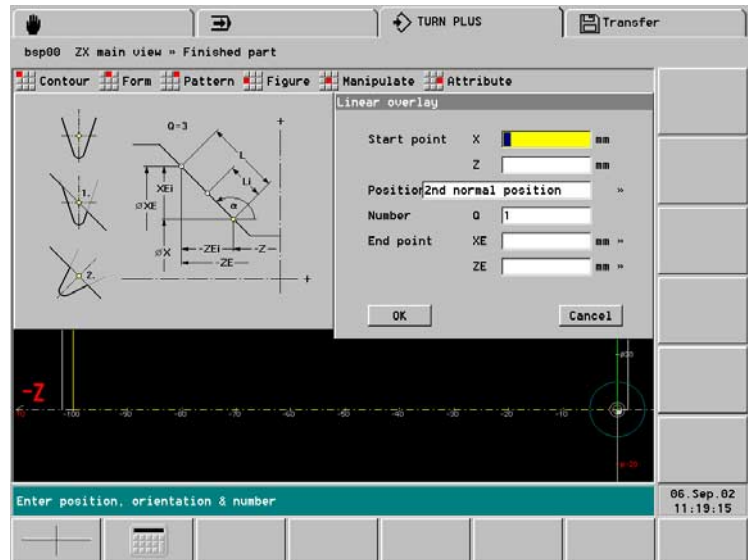
Contour train or last overlay element:

Select "... – Contour"

Select a supporting contour element

Define an overlay (dialog box "Linear/Circular overlay")

TURN PLUS displays the overlay. You can accept (OK) or reject it (cancel).



Integrating a contour train (continued)

If there is more than one solution, select the best one.

TURN PLUS integrates the overlay contours into the existing contour.

See also "6.5.3 Overlay Elements"

6.3.5 Entering Contours Machined with the C Axis

You define standard forms with **Figures**, regular linear or circular figures or holes in **patterns**. To define complex contours, use the **basic elements** "line" and "arc."

Patterns

- Linear hole pattern (drilling)
- Circular hole pattern (drilling)
- Linear figure pattern (milling)
- Circular figure pattern (milling)
- Single bore hole

Figures

- Circle (full circle)
- Rectangle
- Polygon
- Linear groove
- Circular slot

You position patterns and figures on the

- Front face (C-axis machining)
- Lateral surface (C-axis machining)
- Rear side (C-axis machining)

Setting/selecting the reference plane

The reference plane, i.e. the selected window, is marked by a color frame. TURN PLUS refers all activities to this window.

Activate another reference plane (window):

Adjust 1st window configuration

- ▶ "Configuration – Change – Views" (main menu)
- ▶ Mark the window ("window configuration" dialog box)
- ▶ Return to main menu
- ▶ Select "Workpiece - Finished Part."
- ▶ Select the window: "PgUp/PgDn"

Select the 2nd window (reference plane)

- ▶ Select the "turning contour" window
- ▶ Select pattern/figure ("Pattern/Figures")
- ▶ TURN PLUS opens the "Select input plane" dialog box – select the reference plane

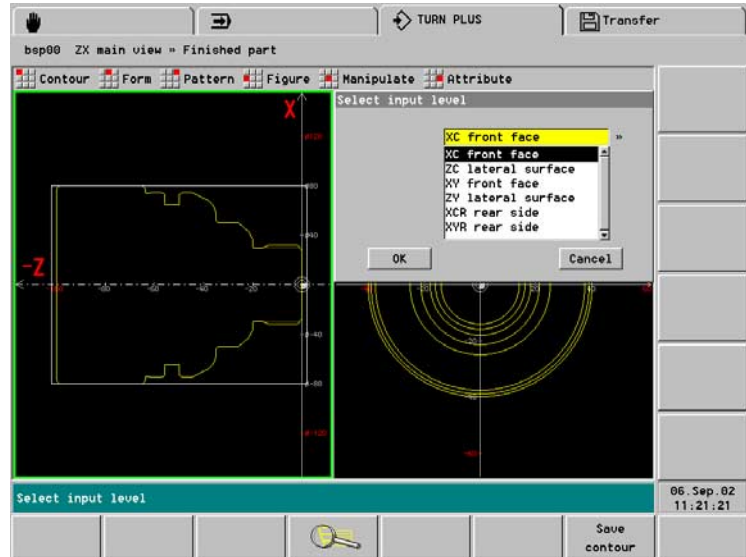
Selection with multiple windows: "PgUp/PgDn"



- Define the complete turning contour before defining the contours for C/Y-axis machining.
- Select the **reference plane** (front face, lateral surface, etc.) before you define the contours for the C/Y axis.

See also

- "6.6.1 Contours of the Front Face and Rear Side"
- "6.6.2 Contours of the Lateral Surface"



Continued ▶

Defining figures

Select "Workpiece – Finished part – Figure"

Select the figure type

If required, adjust the reference plane (front face, lateral surface, etc.)

Select "Machining surface" – Check and correct the "Reference dimension"

- Enter the position
- Press the "figure" button and define the figure

Check your entries; Press OK.

Define pattern/single hole

Select "Workpiece - Finished part - Pattern."

Select pattern/single hole.

If required, adjust the reference plane (front face, lateral surface, etc.)

Select "Machining surface" – Check and correct the "Reference dimension"

Patterns

- Enter pattern positions and data
- Select the Hole/Figure soft key and define the hole/figure

Single hole

- Enter the position
- Press the "hole" button and define the hole

Check your entries; Press OK.

Defining a contour with basic elements

If required, adjust the reference plane (front face, lateral surface, etc.)

Select "Workpiece - Finished part - Contour."

Select "Machining surface" – Check and correct the "Reference dimension"

Define the "Starting point of the contour"

Describe the C/Y contour element for element

After the contour has been completed, press the ESC key twice.

6.3.6 Basics of Operation

Soft keys

You define by soft key the type of dimensions, special function, selection, etc. The following tables and the soft-key overview at the end of this User's Manual illustrate the meaning of the soft keys.

Selection with the touch pad

To make a selection with the touch pad:

- Single selection:
 - ▶ Position cursor on element, etc.
 - ▶ Press left mouse key
- Multiple selection
 - ▶ Switch-on multiple selection by soft key
 - ▶ Position cursor on element, point, etc.
 - ▶ Press the left mouse key
 - ▶ Position cursor next on element, point, etc.
 - ▶ etc.
- Area selection:
 - ▶ Position cursor on the first element
 - ▶ Switch on the area selection by soft key
 - ▶ Position the cursor on the last element
 - Left mouse key: Area selection in direction of contour description
 - Right mouse key: Area selection in direction opposite to contour description

Colors of selection points

- Red: Point marked by cursor, but not selected
- Green: Selected point
- Blue: Point marked by cursor and selected

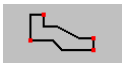


Multiple selection by soft key

Select "Workpiece - Finished part - Form."

Select the desired form element from the drop-down menu.

Place the cursor on "first position"



Switch-on the multiple selection

Mark the selection points in succession



Place cursor on "next position"



Select the marked point



End the selection. Enter the parameters of the form elements



As an alternative you can select all points and deselect the undesired point.

Continued ►

6.3.7 Help Functions for Element Definition

"Delete" drop-down menu

- **Delete "Element/range":** deletes the contour elements last entered.
 - ▶ **Select "Element/range."**
 - ▶ TURN PLUS tags the last element.
 - ▶ Select the contour section by soft key and confirm. The contour section is deleted.
- **Unsolved elements:** deletes all insufficiently defined contour elements immediately.
- **Section:** deletes the entire contour.

"Zero point" drop-down menu

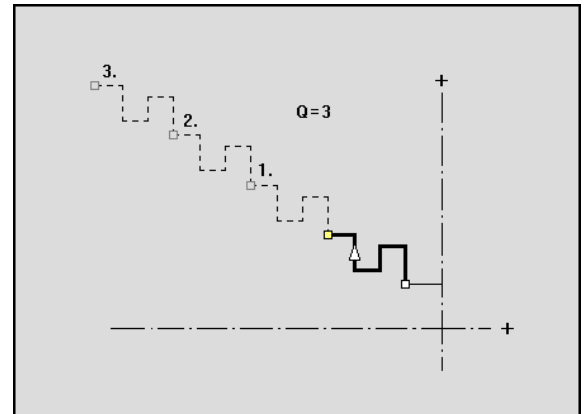
- **Displace:** shifts the zero point of the coordinate system
 - to the position entered (absolute value)
 - by the value entered (incremental value)
- **Reset:** Resets the zero point of the coordinate system to the originally programmed position.

"Duplicate" drop-down menu

- **Series – Linear:** duplicates the selected contour section and adds it n times to the contour.
 - ▶ Select "Series (Row) - Linear."
 - ▶ TURN PLUS tags the last element.
 - ▶ Select the contour section by soft key and confirm.
 - ▶ Enter "Copy in linear series"
 - ▶ TURN PLUS extends the contour.

Parameter ("Copy in linear series")

Q: Number of copies (the contour section is copied Q times)



Continued ►

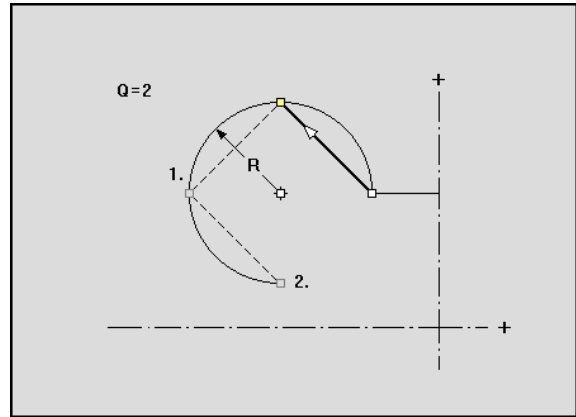
- Series - Circular: duplicates the selected contour section and adds it n times to the contour.
 - ▶ Select "Series (Row) - Circular."
 - ▶ TURN PLUS tags the last element.
 - ▶ Select the contour section by soft key and confirm.
 - ▶ Enter "Copy in circular series" dialog box
 - ▶ TURN PLUS shows a "center of rotation" as a red square – select by soft key the center of rotation and confirm by soft key. TURN PLUS extends the contour.

Parameter ("Copy in circular series")

- Q: Number of copies (the contour section is copied Q times)
 R: Pattern radius

Duplicating circular

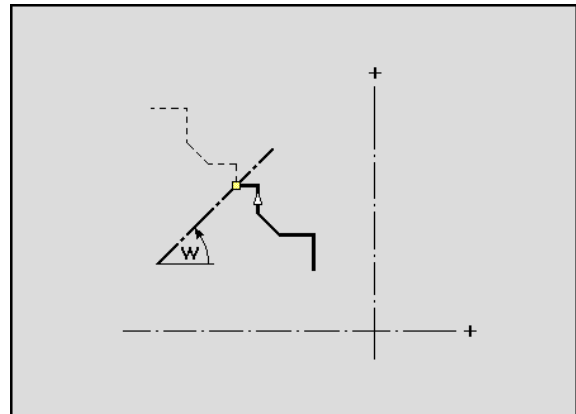
- **Centers of rotation:** TURN PLUS uses the entered radius for creating a circle both around the starting point and the end point of the contour section. The points of intersection of the circles are the two possible centers of rotation.
- The **angle of rotation** is calculated from the distance between starting point and end point of the contour section.
- **Extending the contour:** TURN PLUS duplicates the selected contour section, rotates it and adds it to the contour.



- **Mirroring:** mirrors the selected contour section and adds it to the contour.
 - ▶ Select "Mirroring."
 - ▶ TURN PLUS tags the last element.
 - ▶ Select the contour section by soft key and confirm.
 - ▶ Enter "Copy in circular series" dialog box
 - ▶ Press OK; TURN PLUS extends the contour.

Parameter ("Duplication through mirroring" dialog box)

- W: Angle of the mirror axis – reference for the angle: positive Z axis (the mirror axis runs through the current end point of the contour)



"Info" menu item

The "Info" menu item opens and closes a window containing information on "unsolved geometric elements."

- If the info boxes exceed the window size, move to the next/previous info box by pressing the vertical arrow keys.
- The ALT key calls the parameters of the **last** unsolved element, which can then be edited.

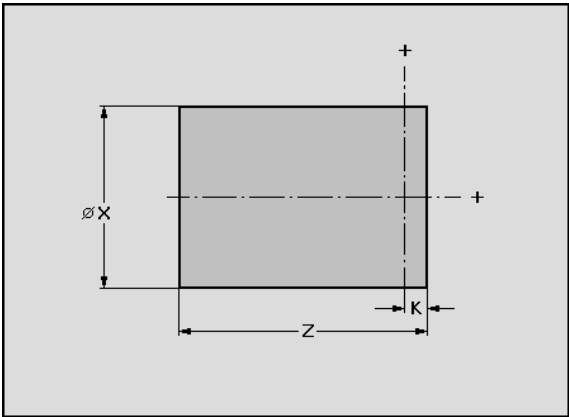
6.4 Contours of Workpiece Blanks

Bar

defines the contour of a cylinder (chuck or bar).

Parameters

- X: ☐ Diameter
 ☐ Diameter of circumference of polygonal blank
- Z: Blank length, including face oversize
- K: Face oversize (distance between workpiece zero point and right edge)

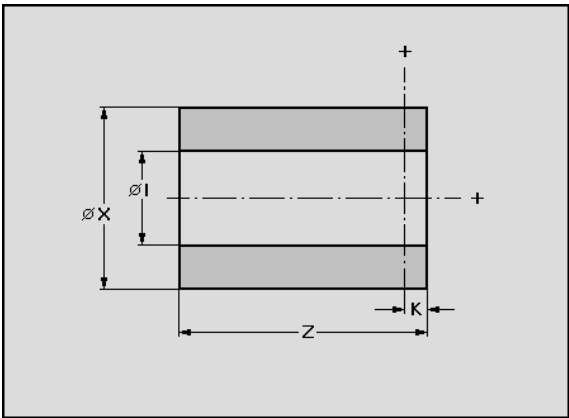


Tube

defines the contour of a hollow cylinder (tube).

Parameters

- X: ☐ Diameter
 ☐ Diameter of circumference of polygonal blank
- I: Inside diameter
- Z: Blank length, including face oversize
- K: Face oversize (distance between workpiece zero point and right edge)



Forging blank (or cast blank)

Generates the workpiece blank from an existing finished part.

Parameters

- Surface: ☐ Cast blank
 ☐ Forging blank
- With bore hole: ☐ Yes
 ☐ No
- K: Equidistant oversize for the complete part
- I: Single allowance (for individual elements or contour sections)



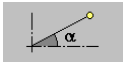
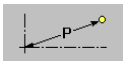
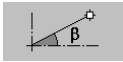
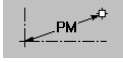


First enter the "single oversize" and then select the contour element/the contour range.

6.5 Contour of Finished Part

6.5.1 Basic Contour Elements

Parameters thatTURN PLUS knows are not requested - the input boxes are locked. Example: On horizontal or vertical lines, only one of the coordinates changes and the angle is defined by the direction of the element.

The set the type of dimensioning by soft key (see table).

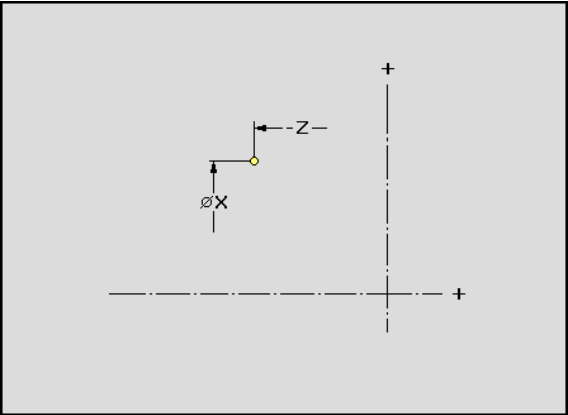
"Dimension of contour elements" soft keys	
	Polar dimensions of the end point: Angle α
	Polar dimensions of the end point: Radius
	Polar dimension of the center: Angle β
	Polar dimension of the center: Radius
	Angle to the predecessor element
	Angle to the successor element

Starting point of contour

To define the starting point, select **Contour**.

Parameters

- X, Z: Starting point of contour
P, α : Starting point of contour in polar coordinates (reference angle α : positive Z axis)



Lines

Use the menu symbol to select the direction of the line and assign it a dimension.

Vertical or horizontal line segments



Select the line direction.



Line at angle



Select the line direction.

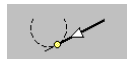


Select a line in any desired direction.

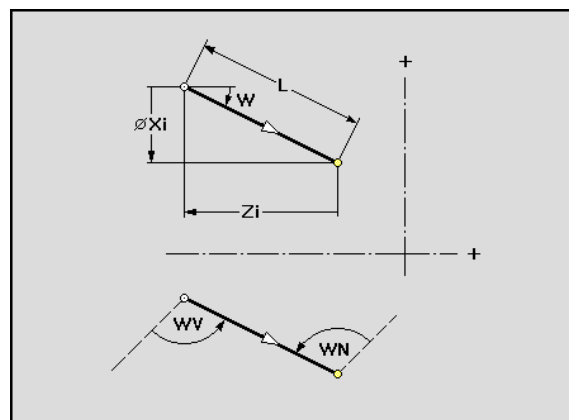
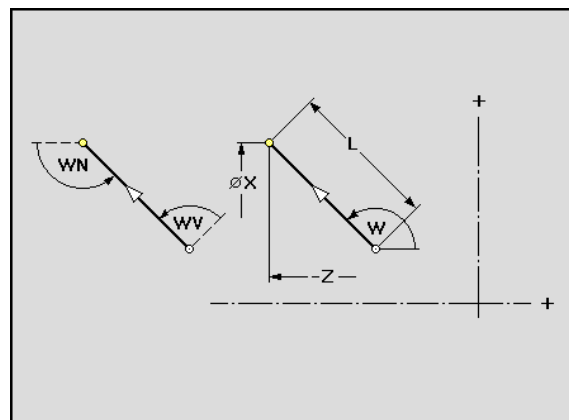
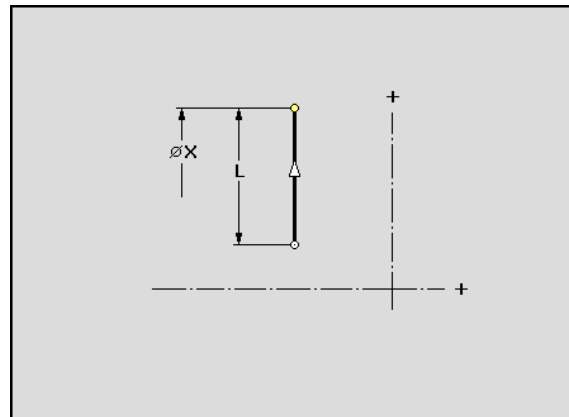
Define the end point of the line and then the transition to the next contour element.

Parameters

- X, Z: End point in Cartesian coordinates
- Xi, Zi: Distance from starting point to end point
- P, α : End point in polar coordinates (reference angle α : positive Z-axis)
- W: Angle of the line (for reference see illustration)
- WV: Angle to the preceding element
- WN: Angle to the successor element
- WV, WN:
 - The angle leads from the preceding/succeeding element counterclockwise to the new element.
 - Arc as preceding/succeeding element: Angle to tangent.
- L: Length of line



Tangential/nontangential: Specify the transition to the next contour element



Arc

Use the menu symbol to select the direction of arc rotation and give the arc a dimension.

The set the type of dimensioning by soft key (see table).

Arc



Select the direction of rotation.

Parameters "end point of arc"

- X, Z: End point in Cartesian coordinates
- Xi, Zi: Distance from starting point to end point
- P, α : End point in polar coordinates (reference angle α : positive Z-axis)
- Pi, α_i : End point polar, incremental (Pi: Linear distance from starting to end point; reference of α_i : see illustration)

Parameters "center of arc"

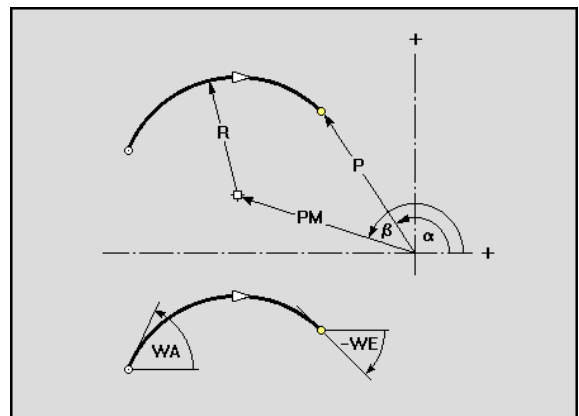
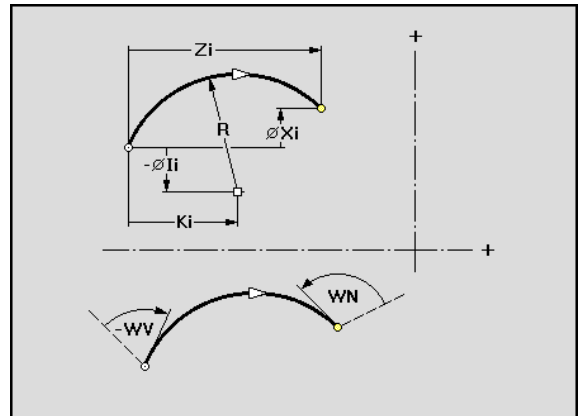
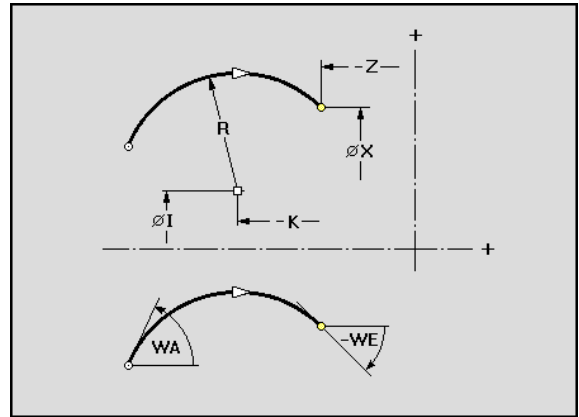
- I, K: Center (XM radius dimension)
- Ii, Ki: Distance from starting point to center point
- PM, β : Center in polar coordinates (reference angle β : positive Z-axis)
- PMi, β_i : Center polar, incremental (PMi: Linear distance from starting point to center; Reference of β_i : Angle between an imaginary line intersecting the starting point and parallel to the Z-axis, and another line from the starting point to the center)

Other parameters

- R: Arc radius
-  Tangential/nontangential: Specify the transition to the next contour element

"Angle" parameters

- WA: Angle between positive Z-axis and tangent in starting point of arc
- WE: Angle between positive Z-axis and tangent in end point of arc
- WV: Angle between preceding element and tangent in the starting point of the arc
- WN: Angle between tangent in arc end point and following element
- WV, WN:
 - The angle leads from the preceding/succeeding element counterclockwise to the new element.
 - Arc as preceding/succeeding element: Angle to tangent.

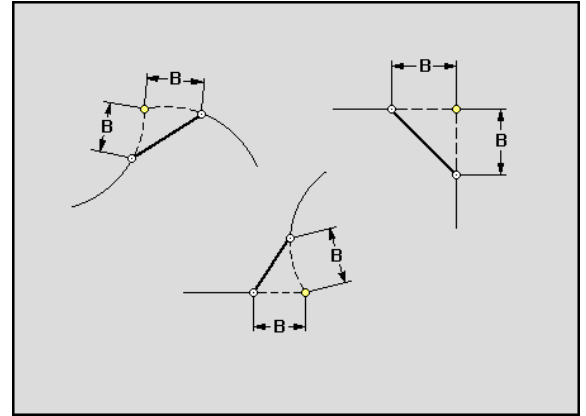


6.5.2 Form elements

Chamfer

Parameters

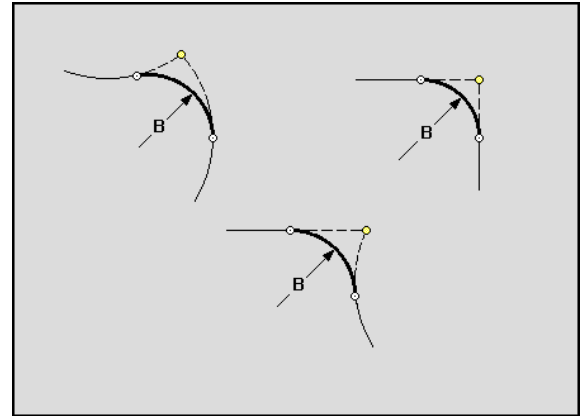
B: Chamfer width



Rounding

Parameters

B: Rounding radius



Undercut type E

Parameters

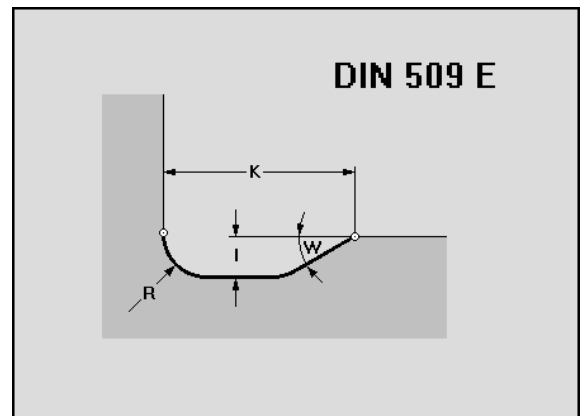
K: Undercut length

I: Undercut depth (radius)

R: Undercut radius (in both corners of the undercut)

W: Approach angle (undercut angle)

TURN PLUS suggests undercut parameters calculated from the diameter (see "11.1.2 Undercut Parameters DIN 509 E").

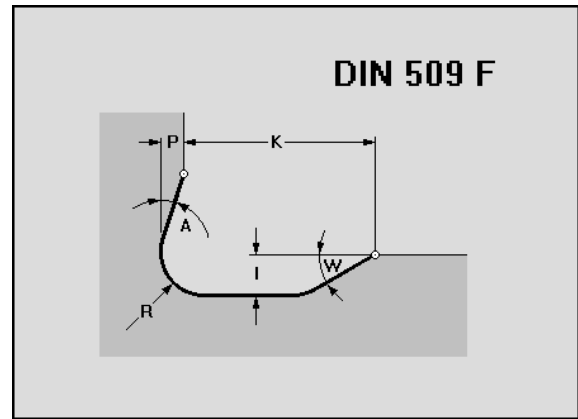


Undercut type F

Parameters

- K: Undercut length
- I: Undercut depth (radius)
- R: Undercut radius (in both corners of the undercut)
- P: transverse depth
- W: Approach angle (undercut angle)
- A: Runout angle (transverse angle)

TURN PLUS suggests undercut parameters calculated from the diameter (see "11.1.3 Undercut Parameters DIN 509 F").



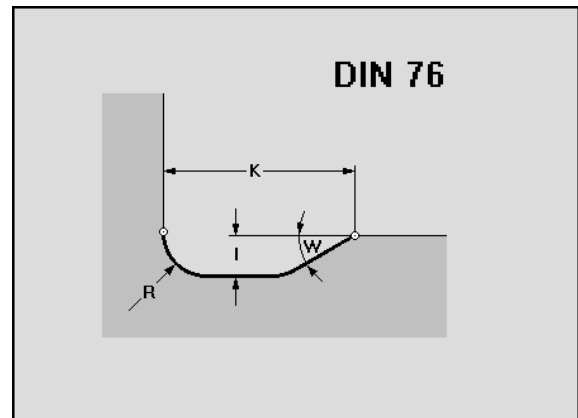
Undercut type G

TURN PLUS proposes the parameter values - but you can overwrite them. The proposed values are based on the metric ISO thread (DIN 13) that is found from the diameter.

- Parameter: see "11.1.1 Undercut Parameters DIN 76"
- Finding the thread pitch: see "11.1.5 Thread Pitch"

Parameters

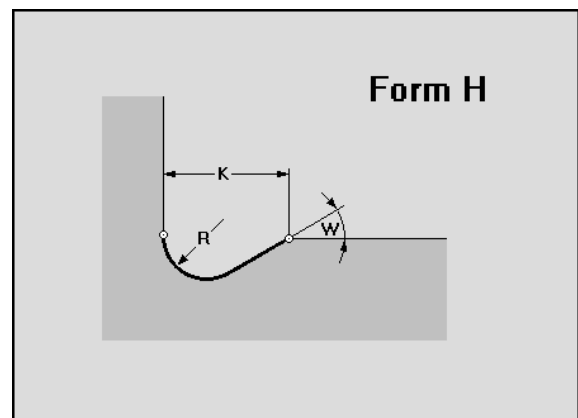
- F: Thread pitch
- K: Undercut length (undercut width)
- I: Undercut depth (radius)
- R: Undercut radius (in both corners of the undercut) – default: $R=0.6 \cdot I$
- W: Approach angle (undercut angle)



Undercut type H

Parameters

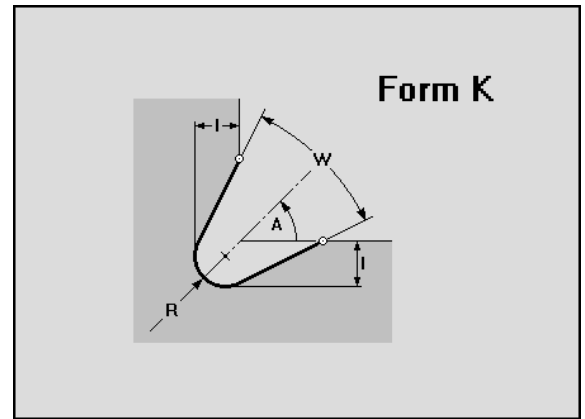
- K: Undercut length
- R: Undercut radius
- W: Approach angle



Undercut type K

Parameters

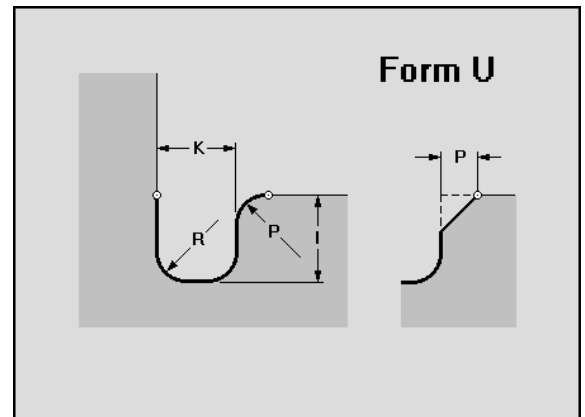
- I: Undercut depth
R: Undercut radius
W: Aperture angle
A: Approach angle (angle to linear axis) – default: 45°



Undercut type U

Parameters

- K: Undercut length (undercut width)
I: Undercut depth (radius)
R: Inside radius (in both corners of recess) – default: 0
P: Outside radius/chamfer
■ No: no chamfer/rounding
■ Chamfer: P = width of chamfer
■ Rounding: P = radius of rounding

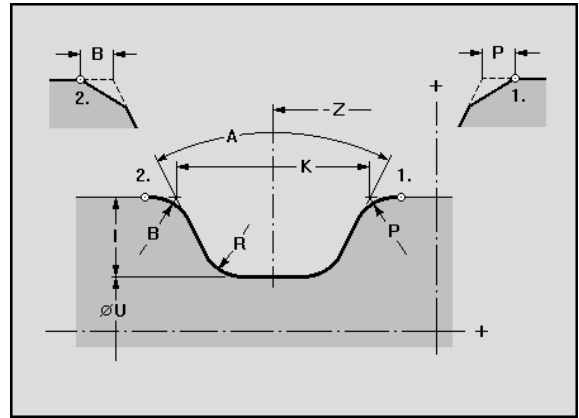


Recess general

Recess general defines an axial or radial recess on a linear reference element. The recess is assigned to the selected reference element.

Parameters

- X/Z: Reference point
K: Recess width (without chamfer/rounding)
l: Recess depth
U: Diameter/radius of recess base (for recesses parallel to the Z-axis)
A: Recess angle (angle between recess edges) – $0^\circ + A < 180^\circ$
P: Outside radius/chamfer (corner far from starting point)
■ No: no chamfer/rounding
■ Chamfer: P = width of chamfer
■ Rounding: P = radius of rounding
B: Outside radius/chamfer (corner near starting point)
■ No: no chamfer/rounding
■ Chamfer: B = width of chamfer
■ Rounding: B = radius of rounding
R: Radius at bottom (inside radius in both recess corners)



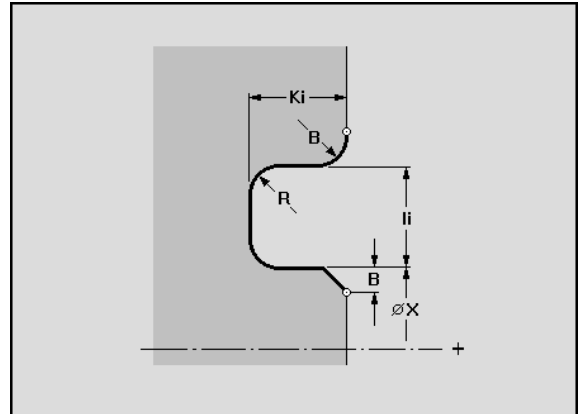
The CNC PILOT refers the recess depth to the reference element. The recess base runs parallel to the reference element.

Recess type D (sealing ring)

This recess type defines an axial or radial recess on the outside or inside of the contour. The recess is assigned to the previously selected reference element.

Parameters

- X: Starting point for radial recess
Z: Starting point for axial recess
l: Diameter/radius of recess base
li: ■ Axial recess: Recess depth
■ Radial recess: Recess width (pay attention to sign!)
Ki: ■ Axial recess: Recess width (pay attention to sign!)
■ Radial recess: Recess depth
B: Outside radius/chamfer (at both recess sides)
■ No: no chamfer/rounding
■ Chamfer: B = width of chamfer
■ Rounding: B = radius of rounding
R: Radius at bottom (inside radius in both recess corners)



Relief turn (form FD)

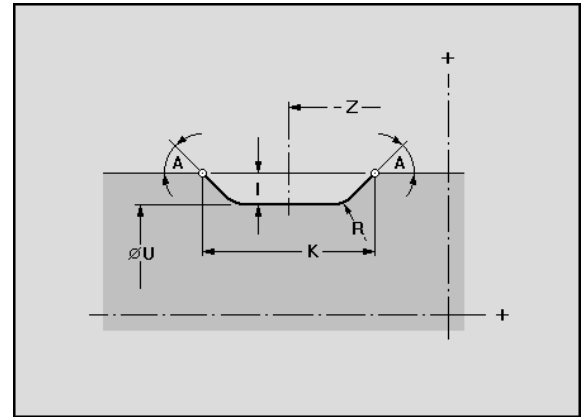
This recess type defines an axial or radial relief turn on a linear reference element. The relief turn is assigned to the previously selected reference element.

Parameters

- X/Z: Reference point
K: Recess width
I: Recess depth
U: Diameter/radius of recess base (provided that the recess base is parallel to the Z-axis)
A: Recess angle ($0^\circ < A \leq 90^\circ$)
R: Inside radius in both corners of the recess



The CNC PILOT refers the recess depth to the reference element. The recess base runs parallel to the reference element.

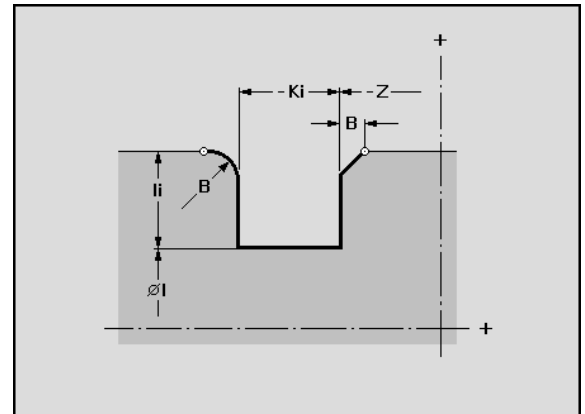


Recess type S (guarding ring)

This recess type defines an axial recess on the outside or inside of the contour. The recess is assigned to the previously selected reference element.

Parameters

- Z: Starting point of recess
Ki: Recess width (pay attention to sign!)
I: Diameter/radius of recess base
li: Recess depth
B: Outside radius/chamfer (at both recess sides)
■ No: no chamfer/rounding
■ Chamfer: B = width of chamfer
■ Rounding: B = radius of rounding

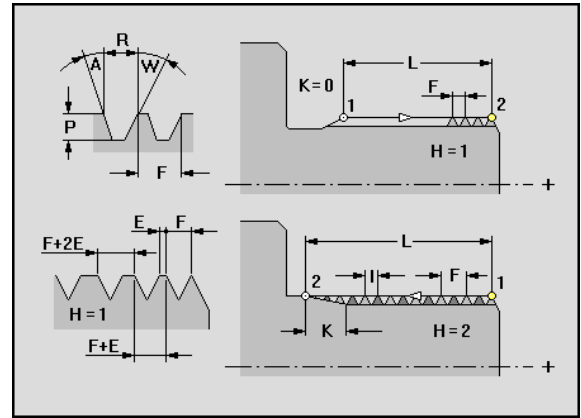


Threads

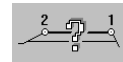
Thread defines the different types of thread.

Parameters

- Q: Type of thread
- Metric ISO fine-pitch thread (DIN 13 Part 2, Series 1)
 - Metric ISO thread (DIN 13 Part 1, Series 1)
 - Metric ISO taper thread (DIN 158)
 - Metric ISO tapered fine-pitch thread (DIN 158)
 - Metric ISO trapezoidal thread (DIN 103 Part 2, Series 1)
 - Flat metric trapezoidal thread (DIN 380 Part 2, Series 1)
 - Metric buttress thread (DIN 513 Part 2, Series 1)
 - Cylindrical round thread (DIN 405 Part 1, Series 1)
 - Cylindrical Whitworth thread (DIN 11)
 - Tapered Whitworth thread (DIN 2999)
 - Whitworth pipe thread (DIN 259)
 - Nonstandard thread
 - UNC US coarse thread
 - UNF US fine thread
 - UNEF US extra fine thread
 - NPT US taper pipe thread
 - NPTF US taper dryseal pipe thread
 - NPSC US cylindrical pipe thread with lubricant
 - US cylindrical pipe thread without lubricant
- V: Direction of rotation
- Right-hand thread
 - Left-hand thread
- D: Select reference point
- Beginning of thread at starting point of element
 - Beginning of thread at end point of element
- F: Thread pitch
- Threads/unit
- The thread pitch/threads per unit must be specified for the "metric fine-pitch thread, tapered thread and tapered fine-pitch thread, trapezoid thread and flat trapezoid thread" as well as for the "nonstandard thread." This parameter may be omitted for the other types of threads. The thread pitch is then calculated from the diameter (see section "11.1.5 Thread Pitch").
- E: Variable pitch (increases/reduces the pitch per revolution by E) – default: 0
- L: Length of the thread (including runout length)
- K: Runout length (for threads without undercut)
- I: Division for determining the threads/unit
- H: Number of thread turns – default: 1
- A, W: Thread angle left/right – with nonstandard thread
- P: Thread depth – with nonstandard thread
- R: Thread width – with nonstandard thread



"Thread" soft keys



Define the direction of the thread

inch

"Threads/inch" is entered instead of "thread pitch"



- Enter either "I" or "H". The rule is: Thread pitch / graduation = number of starts.
- You can assign further **attributes** to the thread (see "6.9.6 Machining Attributes").
- Use the "nonstandard thread" if you want to use individual parameters.



Caution: Danger of collision!

The thread is generated to the length of the reference element. For the machining of threads without an undercut, it is necessary to program the "runout length K" so that the overrun can be executed by the CNC PILOT without danger of collision.

(Centric) bore hole

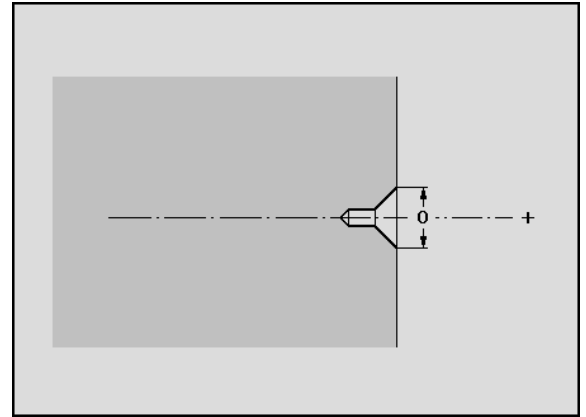
Bore hole defines a single bore hole **on the turning center** (front or back).

The hole can contain the following elements:

- Centering
- Core drilling
- Countersinking
- Thread

Centering parameters

O: Centering diameter



Core hole parameter

B: Hole diameter

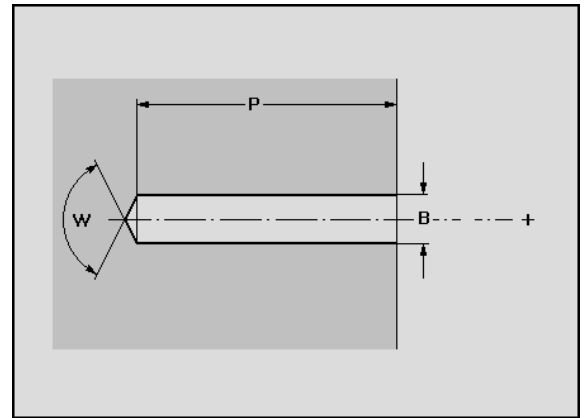
P: Depth of hole (excluding point)

W: Point angle

■ $W=0^\circ$: "Feed rate reduction ($V=1$)"

■ $W>0^\circ$: Point angle

Fit: H6...H13 or "none" (see "6.16.6 Drilling")

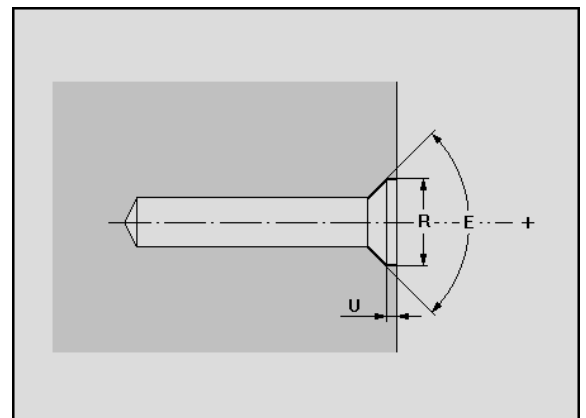


Countersinking parameter

R: Countersinking diameter

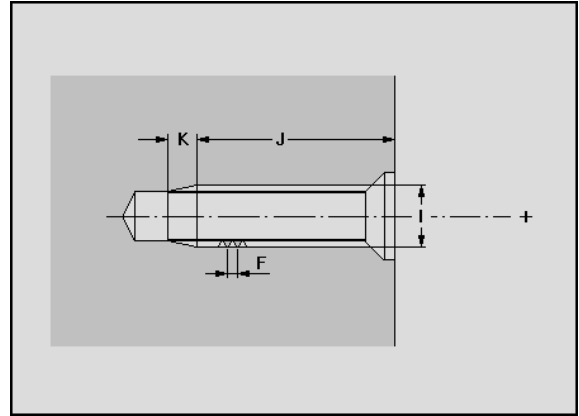
U: Countersinking depth

E: Countersinking angle



Thread parameters

I: Nominal diameter
 J: Thread depth
 K: Thread runout length
 F: Thread pitch
 Thread type: Right-hand/left-hand



6.5.3 Overlay Elements

Call: Menu item "Form - Form element - ..." (submenu "Finished part")

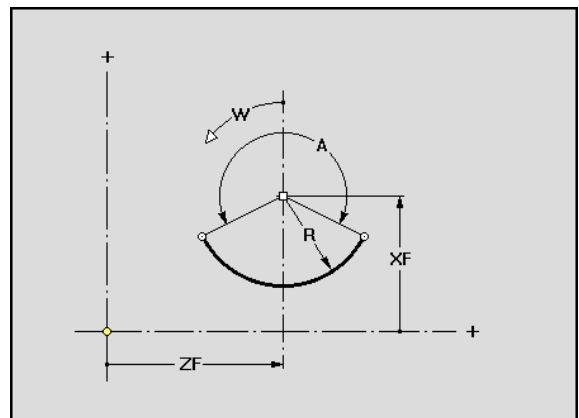
- Select the contour trains arc, wedge or pontoon, define the element and superimpose it after the definition.
- With the menu item "Form – Form element – Contour," TURN PLUS superimposes the last loaded contour train. This is either the previously loaded contour train (main menu: "Program – Load – Contour train") or the last defined overlay element.

Circular arc

Circle center as reference.

Parameters

XF, ZF: Reference point shift
 R: Arc radius
 A: Aperture angle
 W: Angle of rotation: The overlay contour is rotated by the "angle of rotation."



Wedge/rounded wedge

Reference point: Wedge tip/midpoint of rounding

Parameters

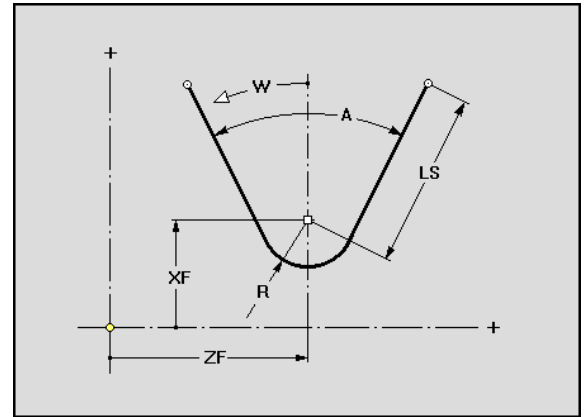
XF, ZF: Reference point shift

R: ☐ R>0: Radius of rounding
☐ R=0: No rounding

A: Aperture angle

LS: Length of wedge sides (projecting element parts are cut at the "points of overlay")

W: Angle of rotation: The overlay contour is rotated by the "angle of rotation."



Ponton

Reference point: Center of base element

Parameters

XF, ZF: Reference point shift

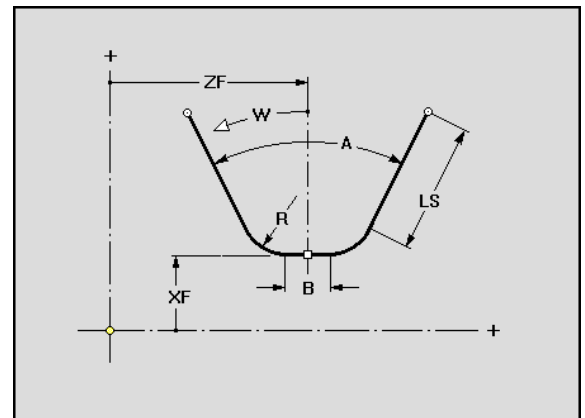
R: ☐ R>0: Radius of rounding
☐ R=0: No rounding

A: Aperture angle

LS: Length of ponton sides (projecting element parts are cut at the "points of overlay")

B: Width of base element

W: Angle of rotation: The overlay contour is rotated by the "angle of rotation."



Superimposition ("Overlay")

Depending on the form of the supporting contour elements, there is a

- ☐ Linear overlay or
- ☐ Circular superimposition

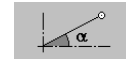


The overlay positions can deviate from the supporting contour element.

"Linear overlay" soft keys

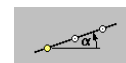


Enter the length (instead of end point)

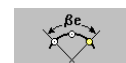


Enter the length (instead of end point)

Circular overlay soft keys



Define the first overlay position by angle



Define the last overlay position by angle

Continued ►

"Linear overlay" parameters

X, Z: Starting point – Position of the first overlay element

Position: ■ Original position: Inserts the "original" overlay contour in the supporting contour (see Help graphic "1.").

■ Normal position: Rotates the overlay contour about the pitch angle of the supporting contour element and inserts it then in the supporting contour (see help graphic "2.").

Q: Number of overlay elements

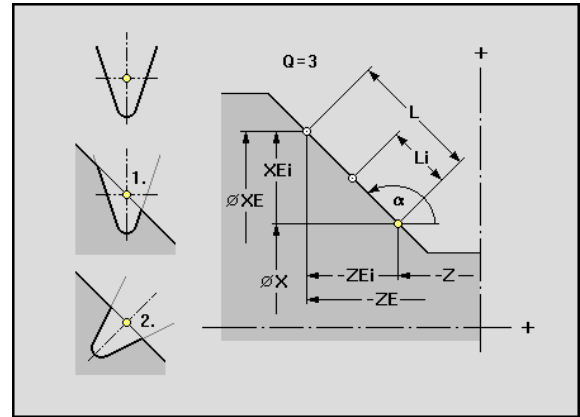
XE, ZE: End point – Position of the last overlay element

XEi, ZEi: Incremental end point

L: Distance between the first and last overlay element

Li: Distance between the overlay elements

α : Angle – default: Angle of the supporting contour element



"Circular overlay" parameters

X, Z: Starting point – Position of the first overlay element

α : Starting point as angle (reference: one line running parallel to the Z axis and through the center of the selected arc)

Position: ■ Original position: Inserts the overlay contour "as it is" into the support contour (see section 1. in the figure).

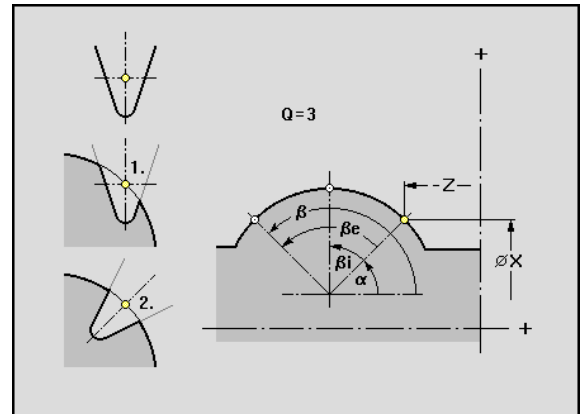
■ Normal position: rotates the overlay contour about the normal position: The overlay contour is rotated by the angle of the "overlay point" and is inserted into the support contour (see section 2. in the figure).

Q: Number of overlay elements

β : End point - position of the last element to be superimposed (reference: a line segment running parallel to the Z axis and intersecting the midpoint of the selected arc)

β_e : Angle between the first and last overlay element

β_i : Angle between the individual overlay elements



The **direction of rotation** according to which the overlay contours are arranged corresponds to the direction of rotation of the supporting contour element.



The reference point of the contour to be superimposed is positioned on the "point of overlay."

6.6 C-Axis Contours

6.6.1 Contours on the Front and Rear Face

Milling depth

For figures, enter the parameter "Depth P" If you use individual elements for describing a milling contour,TURN PLUS opens the "Pocket/contour" dialog box after you have finished your entries, which requests "Depth P."

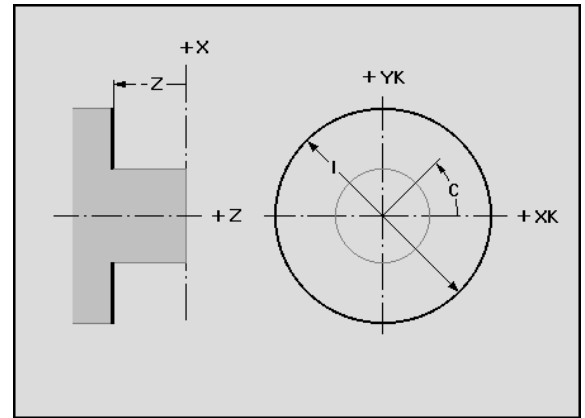
"Depth P > 0" defines a "pocket."

Position of end-face contours

TURN PLUS takes the selected reference plane and suggests it as reference dimension.

"Reference data" dialog box

Z: Reference dimension



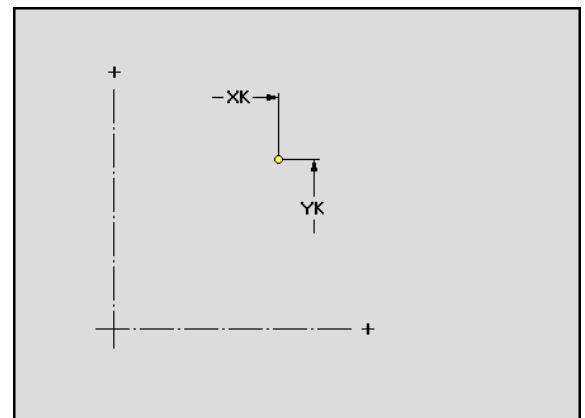
Starting point of end-face contour

To define the starting point, select **Contour**.

Parameters

XK,YK: Starting point of the contour in Cartesian coordinates

P, α : Starting point of contour in polar coordinates (reference angle α : positive XK axis)



Line segment in end-face contour

Use the menu symbol to select the direction of the line and assign it a dimension.

Parameters

XK,YK: End point in Cartesian coordinates

XKi,YKi: Distance from starting point to end point.

P, α : End point in polar coordinates (reference angle α : positive XK axis)

W: Angle of the line (for reference see illustration)

WV: Angle to the preceding element

WN: Angle to the successor element

WV, WN:

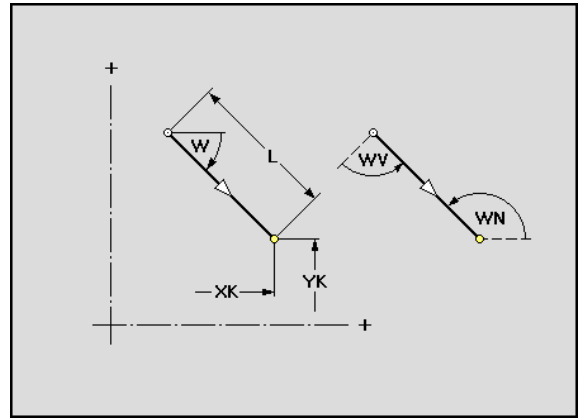
■ The angle leads from the preceding/succeeding element counterclockwise to the new element.

■ Arc as preceding/succeeding element: Angle to tangent.

L: Length of line



Tangential/nontangential: Specify the transition to the next contour element



Circular arc in end-face contours

Use the menu symbol to select the direction of arc rotation and give the arc a dimension.

Parameters "end point of arc"

XK,YK: End point in Cartesian coordinates

XKi,YKi: Distance from starting point to end point

P, α : End point in polar coordinates (reference angle α : positive XK axis)

Pi, α_i : End point polar, incremental. Pi: Linear distance from starting to end point; Reference of α_i : Angle between an imaginary line intersecting the starting point and parallel to the XK axis, and another line from the starting point to the end point.

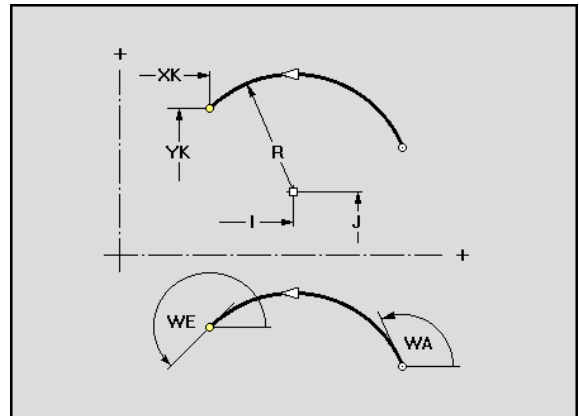
Parameters "center of arc"

I, J: Center in Cartesian coordinates

Ii, Ji: Difference from starting point to center point in XK,YK direction

β , PM: Center in polar coordinates (reference angle β : positive XK axis)

β_i , PMi: Center polar, incremental (PMi: linear distance from starting point to center; reference of β_i : angle between imaginary line intersecting the starting point and parallel to the XK-axis, and another line from the starting point to the center)

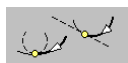


End point must not be the starting point (no full circle).

Continued ►

Other parameters

R: Arc radius



Tangential/nontangential: Specify the transition to the next contour element

"Angle" parameters

WA: Angle between positive XK-axis and tangent in starting point of arc

WE: Angle between positive XK-axis and tangent in end point of arc

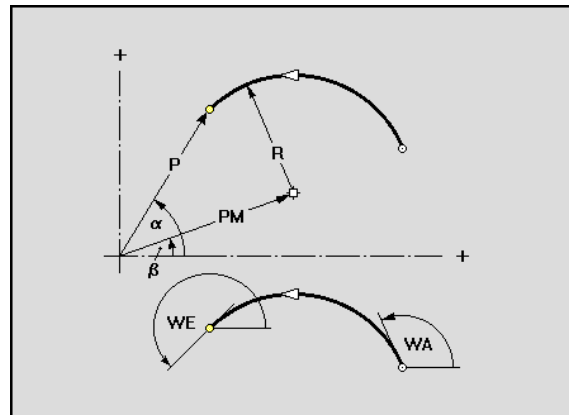
WV: Angle between preceding element and tangent in the starting point of the arc

WN: Angle between tangent in arc end point and following element

WV, WN:

■ The angle leads from the preceding/succeeding element counterclockwise to the new element.

■ Arc as preceding/succeeding element: Angle to tangent.



Single hole

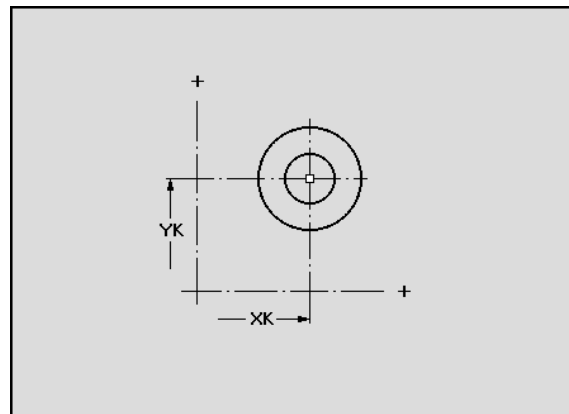
"Reference point" parameters

XK, YK: Hole center in Cartesian coordinates

α , PM: Center of hole in polar coordinates (reference angle α : positive XK-axis)

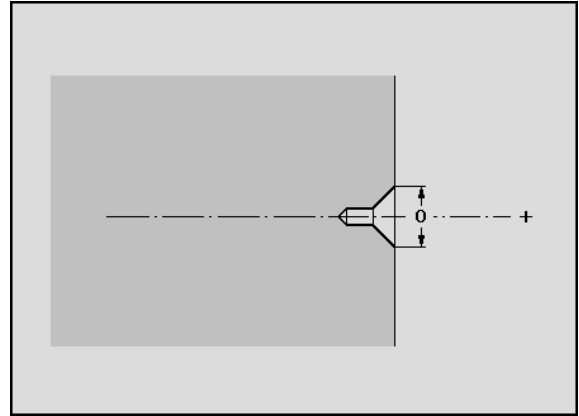
The hole can contain the following elements:

- Centering
- Core drilling
- Countersinking
- Thread



Centering parameters

O: Centering diameter



Core hole parameters

B: Hole diameter

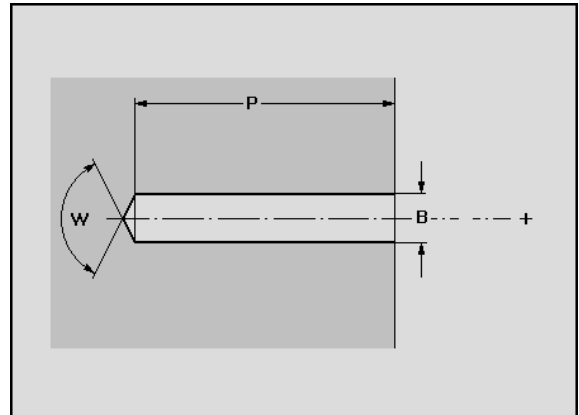
P: Depth of hole (excluding point)

W: Point angle

■ $W=0^\circ$: "Feed rate reduction ($V=1$)"

■ $W>0^\circ$: Point angle

Fit: H6...H13 or "none" (see "6.16.6 Drilling")

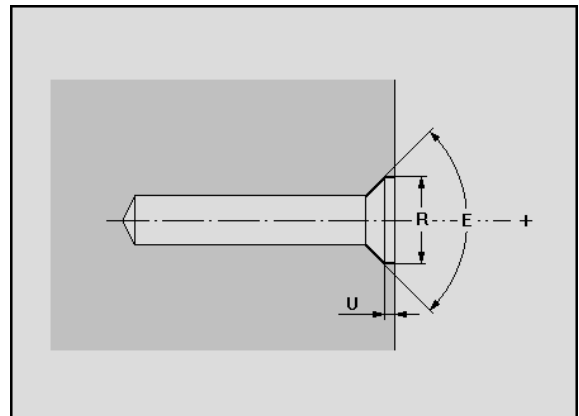


Countersinking parameter

R: Countersinking diameter

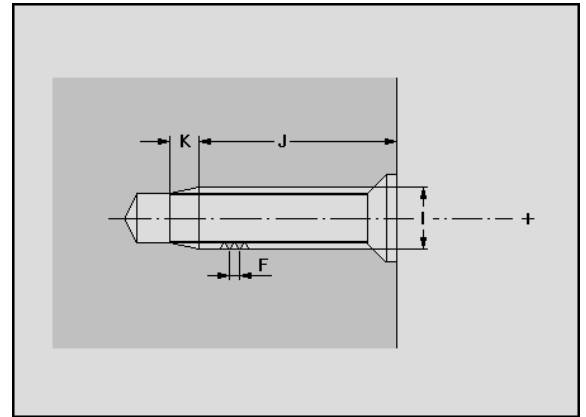
U: Countersinking depth

E: Countersinking angle



"Thread" parameters

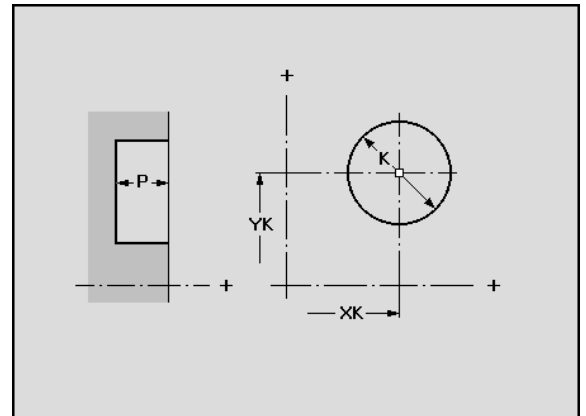
I: Nominal diameter
 J: Thread depth
 K: Thread runout length
 F: Thread pitch
 Thread type: Right-hand/left-hand



Circle (full circle)

Parameters

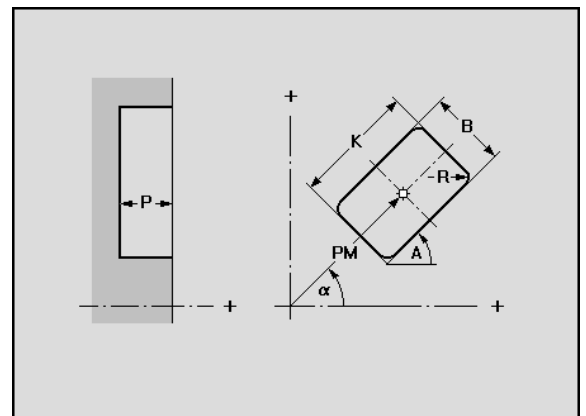
XK, YK: Center in Cartesian coordinates
 α , PM: Center in polar coordinates (reference angle α : positive XK-axis)
 R/K: Radius/diameter of circle
 P: Depth of the figure



Rectangle

Parameters

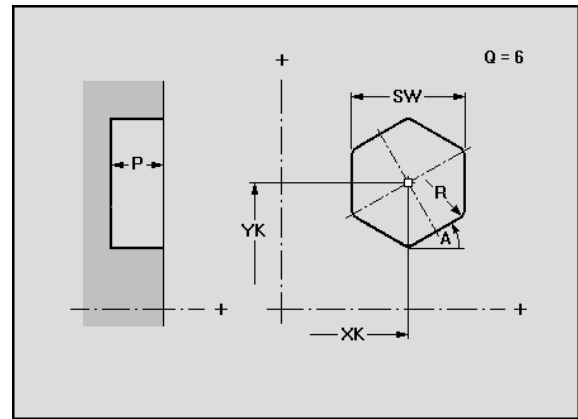
XK, YK: Center in Cartesian coordinates
 α , PM: Center in polar coordinates (reference angle α : positive XK-axis)
 A: Angle to longitudinal axis of rectangle (reference: XK-axis)
 K: Length of rectangle
 B: Width of rectangle
 R: Chamfer/rounding
 ■ Width of chamfer
 ■ Radius of rounding
 P: Depth of the figure



Polygon

Parameters

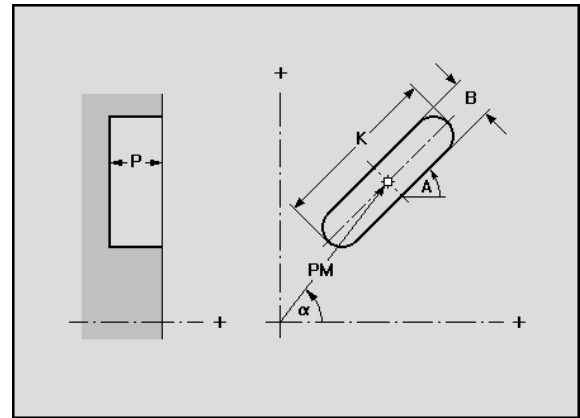
- XK,YK: Center in Cartesian coordinates
 α , PM: Center in polar coordinates (reference angle α : positive XK-axis)
A: Angle to a polygon side (reference: XK-axis)
Q: Corner number ($Q \geq 3$)
K: Length of side
SW: Width across flats (inscribed circle diameter)
R: Chamfer/rounding
■ Width of chamfer
■ Radius of rounding
P: Depth of the figure



Linear slot

Parameters

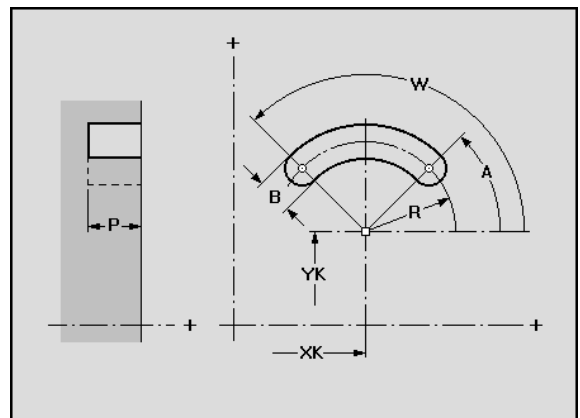
- XK,YK: Center in Cartesian coordinates
 α , PM: Center in polar coordinates (reference angle α : positive XK-axis)
A: Angle of longitudinal axis of slot (reference: XK-axis)
K: Slot length
B: Slot width
P: Depth of the figure



Circular slot

Parameters

- XK,YK: Center of curvature in Cartesian coordinates
 α , PM: Center of curvature in polar coordinates (reference angle α : positive XK-axis)
A: Starting angle (starting point) of the slot (reference: XK axis)
W: End angle (end point) of the slot (reference: XK axis)
R: Curvature radius (reference: center point path of the slot)
B: Slot width
P: Depth of the figure



Linear Hole Pattern, Linear Figure Pattern

Parameters

XK, YK: Starting point of pattern in Cartesian coordinates

α , P: Starting point of pattern in polar coordinates (reference angle α : positive XK axis)

Q: Number of holes/figures—default: 1

I, J: end point of pattern in Cartesian coordinates

Ii, Ji: Distance between two figures in XK/YK direction

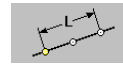
β : Angle to the longitudinal axis of the pattern (reference: XK axis)

L: Total length of pattern

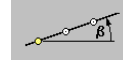
Li: Distance between two figures (pattern distance)

Hole description/figure description

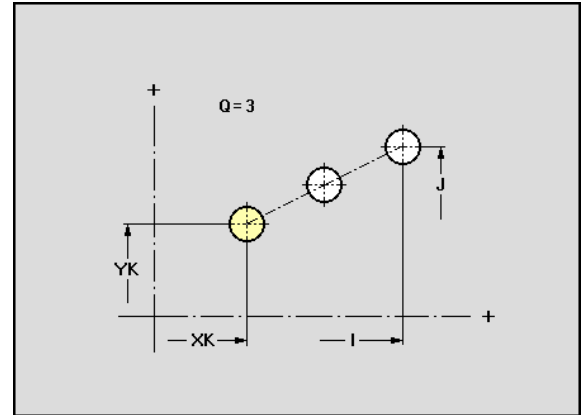
"Type of dimensioning" soft keys



Linear pattern: Enter the length



Linear pattern: Enter the angle



Circular Hole Pattern, Circular Figure Pattern

Parameters

XK, YK: Center of pattern in Cartesian coordinates

α , PM: Center of pattern in polar coordinates (reference angle α : positive XK axis)

Q: Number of figures

Orientation:

- Clockwise n
- Counterclockwise

R/K: Radius/diameter of pattern

A, W: Starting angle, end angle—position of first/last figure (reference: XK axis)—special cases:

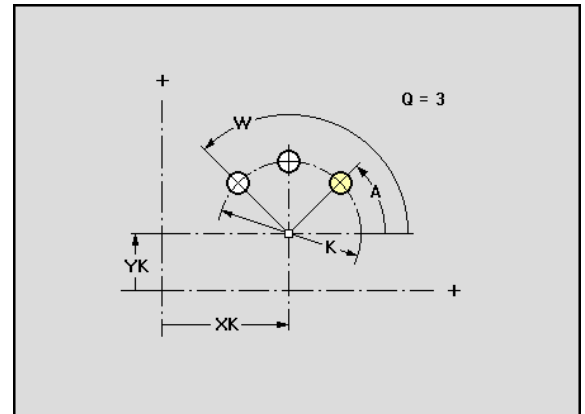
- Without A and W: Full-circle subdivision, beginning with 0°
- Without W: Full-circle subdivision

Wi: Angle between two figures (algebraic sign has no effect)

Position of figures:

- Normal position: The original figure is rotated about the pattern center (rotation about the pattern center)
- Original position: The position of the original figure remains (translation)

Hole description/figure description



In patterns with circular slots, the "center of curvature" is added to the pattern position (see User's Manual, "4.5.8 Circular Pattern with Circular Slots").

6.6.2 Contours of the Lateral Surface

Cartesian or polar dimensions

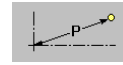
The "linear dimension CY" is given with respect to the unrolled surface under "reference diameter."

Milling depth

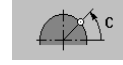
For figures, enter the parameter "Depth P." If you use individual elements for describing a milling contour, TURN PLUS opens the "Pocket/contour" dialog box after you have finished your entries, which requests "Depth P."

"Depth P" > 0 defines a pocket.

"Dimensions of lateral surface" soft key



Polar dimensions



Angle or angle as linear dimension



Polar dimension (parameter "P"):
n "P" is given with respect to the **unrolled lateral surface**.

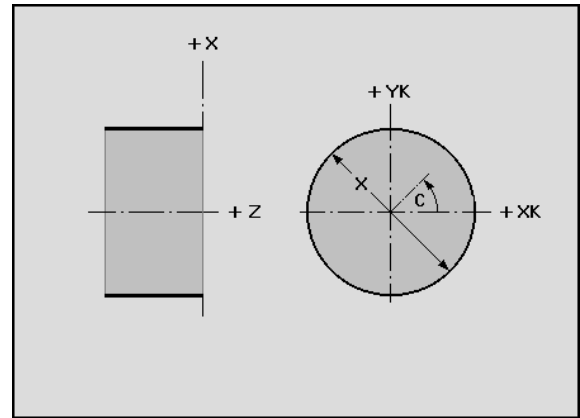
■ Select the desired solution when there are two possibilities.

Position of lateral-surface contours

TURN PLUS takes the selected reference plane and suggests it as reference diameter.

"Reference data" dialog box

X: Reference diameter

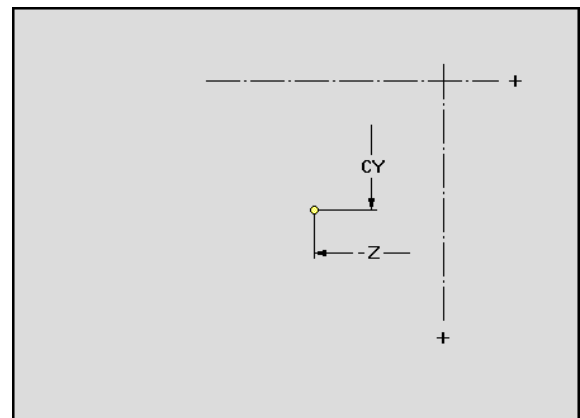


Starting point of lateral surface contour

To define the starting point, select **Contour**.

Parameters

- Z: Starting point of contour
- P: Starting point of contour – polar
- CY: Starting point of contour – angle as linear dimension
- C: Starting point of contour – angle



Line segment in a lateral surface contour

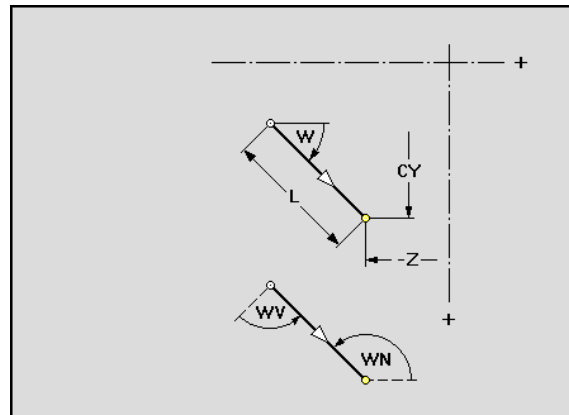
Use the menu symbol to select the direction of the line and assign it a dimension.

Parameters

- Z: End point of line
P: End point of the line - polar
CY: End point of the line - angle as "linear dimension"
C: End point of the line - angle
W: Angle of the line (for reference see illustration)
WV: Angle to the preceding element
WN: Angle to the successor element
WV, WN:
■ The angle leads from the preceding/succeeding element counterclockwise to the new element.
■ Arc as preceding/succeeding element: Angle to tangent.
L: Length of line



Tangential/nontangential: Specify the transition to the next contour element



Circular arc in lateral surface contour

Use the menu symbol to select the direction of arc rotation and give the arc a dimension.

Parameters "end point of arc"

- Z: End point
P: End point - polar
CY: End point - angle as "linear dimension"
C: End point - angle

Center-of-arc parameters

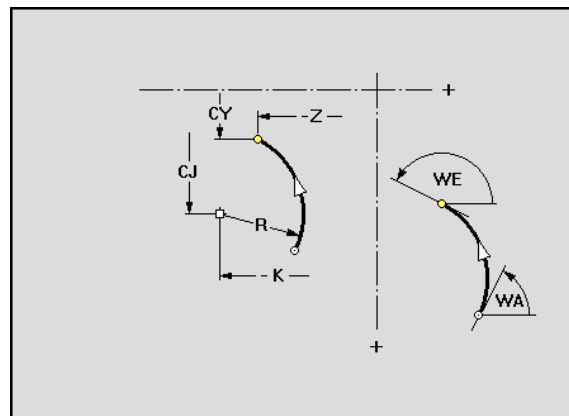
- K: Center
CJ: Center (angle as linear dimension - reference: unrolled cylinder at "reference diameter")
PM: Center point, polar
 β : Center (angle)

Other parameters

- R: Arc radius

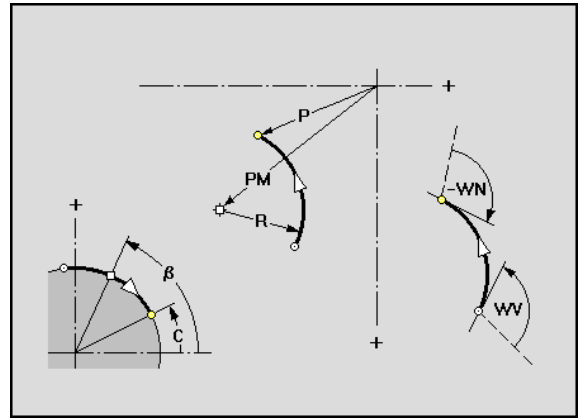


Tangential/nontangential: Specify the transition to the next contour element



Angle parameters

- WA: Angle between positive Z-axis and tangent in starting point of arc
- WE: Angle between positive Z-axis and tangent in end point of arc
- WW: Angle between preceding element and tangent in the starting point of the arc
- WN: Angle between tangent in arc end point and following element
- WV, WN:
- The angle leads from the preceding/succeeding element counterclockwise to the new element.
 - Arc as preceding/succeeding element: Angle to tangent.



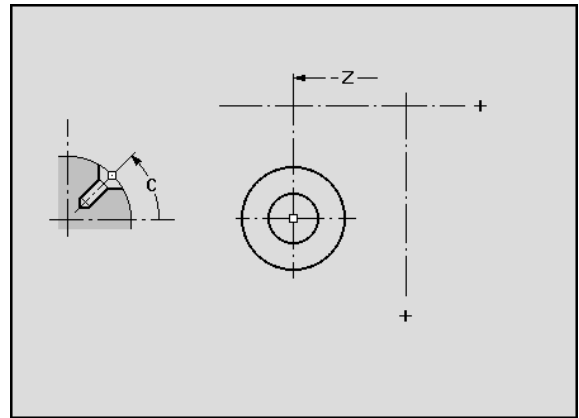
Single hole

"Reference point" parameters

- Z: Center of hole
- CY: Center of the hole - angle as "linear dimension"
- C: Center of the hole - angle

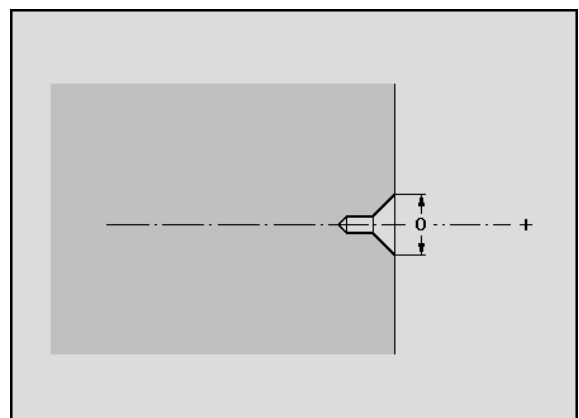
The hole can contain the following elements:

- Centering
- Core drilling
- Countersinking
- Thread



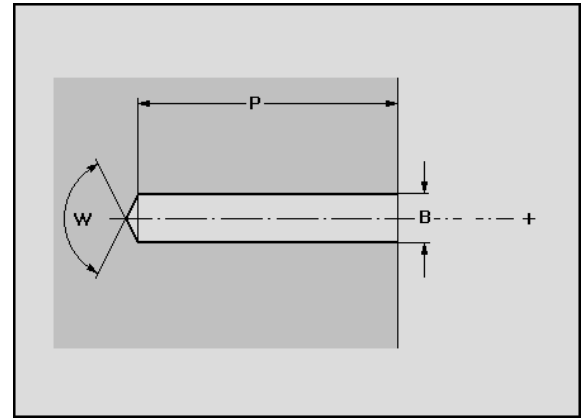
Centering parameters

- O: Centering diameter



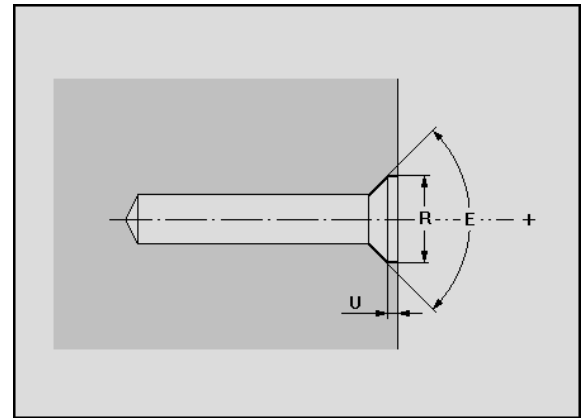
Core hole parameter

- B: Hole diameter
P: Hole depth (depth of hole and countersink - without drill point and center point)
W: Point angle
■ $W=0^\circ$: "Feed rate reduction ($V=1$)"
■ $W>0^\circ$: Point angle
Fit: H6...H13 or "none" (see "6.16.6 Drilling")



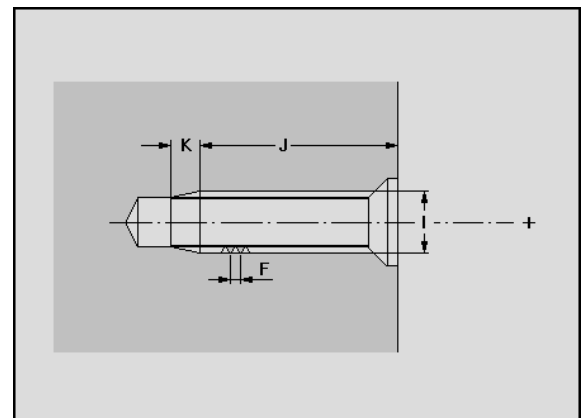
Countersinking parameters

- R: Countersinking diameter
U: Countersinking depth
E: Countersinking angle



Thread parameters

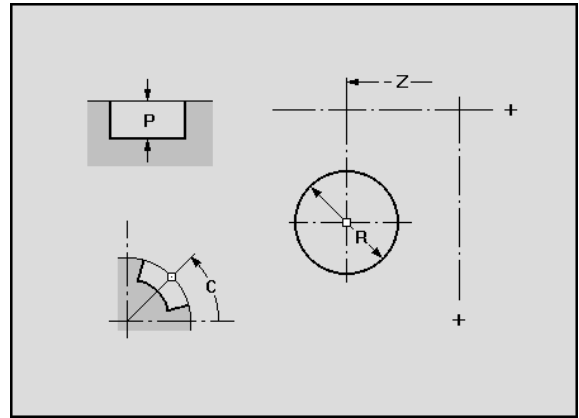
- I: Nominal diameter
J: Thread depth
K: Thread runout length
F: Thread pitch
Thread type: Right-hand/left-hand



Circle (full circle)

Parameters

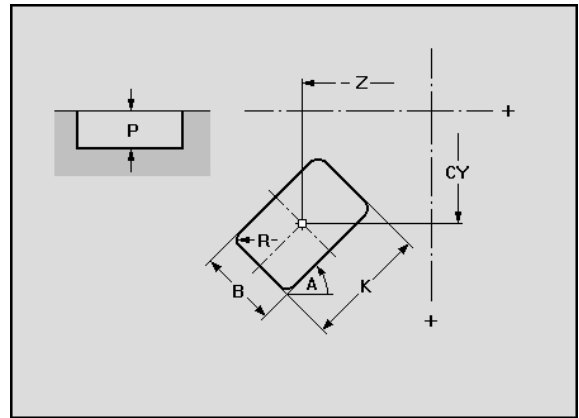
- Z: Center of the figure
 CY: Center of the figure - angle as "linear dimension"
 C: Center of figure - angle
 R: Radius
 K: Diameter of circle
 P: Depth of the figure



Rectangle

Parameters

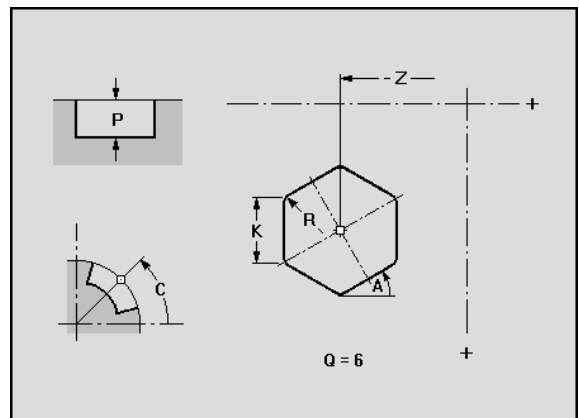
- Z: Center of the figure
 CY: Center of the figure - angle as "linear dimension"
 C: Center of figure - angle
 A: Angle to longitudinal axis of rectangle (reference Z-axis)
 K: Length of rectangle
 B: Width of rectangle
 R: Chamfer/rounding
 ■ Width of chamfer
 ■ Radius of rounding
 P: Depth of the figure



Polygon

Parameters

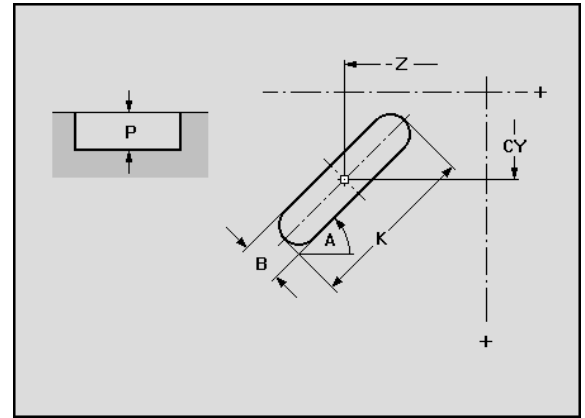
- Z: Center of the figure
 CY: Center of the figure - angle as "linear dimension"
 C: Center of figure - angle
 A: Angle to a polygon side (reference: Z-axis)
 Q: Corner number ($Q \geq 3$)
 K: Length of side
 SW: Width across flats (inscribed circle diameter)
 R: Chamfer/rounding
 ■ Width of chamfer
 ■ Radius of rounding
 P: Depth of the figure



Linear slot

Parameters

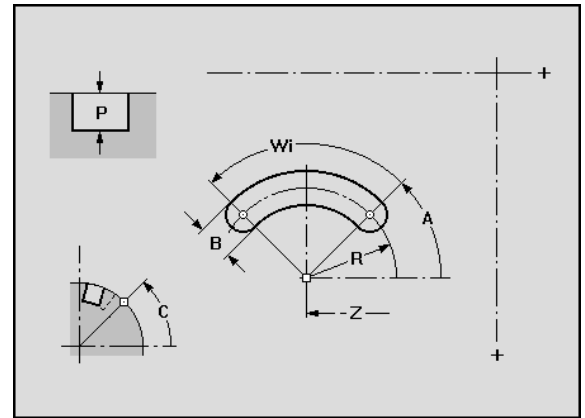
- Z: Center of figure
 CY: Center of the figure - angle as "linear dimension"
 C: Center of figure - angle
 A: Angle to longitudinal axis of slot (reference: Z-axis)
 K: Slot length
 B: Slot width
 P: Depth of the figure



Circular slot

Parameters

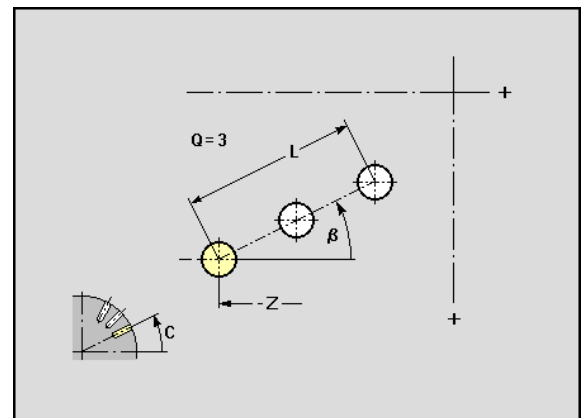
- Z: Center of figure
 CY: Center of the figure - angle as "linear dimension"
 C: Center of figure - angle
 A: Starting angle (starting point) of the slot (reference: Z axis)
 W: End angle (end point) of the slot (reference: Z axis)
 R: Curvature radius (reference: center point path of the slot)
 B: Slot width
 P: Depth of the figure



Linear hole pattern, linear figure pattern

Parameters

- Z: Starting point of pattern
 CY: Starting point of pattern – angle as linear dimension
 C: Starting point of pattern – angle
 Q: Number of figures
 K: End point of pattern
 Ki: Distance between the figures (in Z direction)
 Cye: End point of pattern – angle as "linear dimension"
 CYi: Distance between figures – as "linear dimension"



Continued ►

- L: Total length of pattern
- Li: Distance between the figures (pattern distance)
- β : Angle to the longitudinal axis of the pattern (reference: Z axis)
- W: Ending angle
- Wi: Distance between the figures as angle (pattern distance)

Hole description/figure description



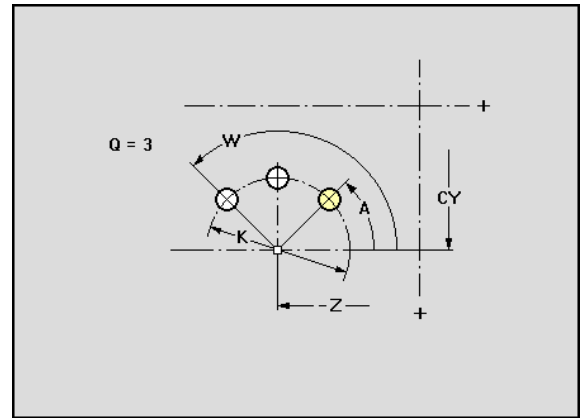
If you do not program the end point, the holes/figures will be distributed evenly along the circumference.

Circular Hole Pattern, Circular Figure Pattern

Parameters

- Z: Center of pattern
 - CY: Center of pattern—angle as linear value
 - C: Center of pattern—angle
 - Q: Number of holes/figures—default: 1
 - Orientation:
 - Clockwise n
 - Counterclockwise
 - R: Pattern radius
 - K: Pattern diameter
 - A, W: Starting angle, end angle—position of first/last figure (reference: XK axis)—special cases:
 - Without A and W: Full-circle subdivision, beginning with 0°
 - Without W: Full-circle subdivision
 - Wi: Angle between two figures (algebraic sign has no effect)
- For figures, except for circles, you must enter the position of the figure in the figure description:
- Normal position (H=0): Initial figure is rotated about the center of the pattern.
 - Original position (H=1): Position of the figure remains the same (translation).

Hole description/figure description



In patterns with circular slots, the “center of curvature” is added to the pattern position (see User’s Manual, “4.5.8 Circular Pattern with Circular Slots”).

6.7 Manipulating Contours

Note when editing contours:

- For contour elements that are superimposed by form elements, the indicated end points or the end points to be entered are given with respect to the "theoretical" end point. When contour elements are modified, the system automatically adjusts chamfers, roundings, threads and undercuts to the new position.
- Sequence, starting point and end point of a contour element are determined by the direction of definition.
- After trimming, deleting or inserting operations have been executed, TURN PLUS analyzes whether successive elements can be combined to form a line segment or an arc. The edited contour is **standardized**.



The turning contour cannot be edited if contours machined with the C or Y axis exist.

6.7.1 Editing the Contours of a Blank Part

If there is a **standard blank** (bar, tube) you can:

- **Delete(d)** by selecting: "Workpiece – Blank – Manipulate - Delete – Contour"
 - **Disconnect** by selecting: "Workpiece - Blank - Manipulate - Disconnect"
- The standard blank is resolved into individual contour elements. You can then manipulate the individual elements.

If there is a **casting** or a **blank with individual elements** was defined, manipulate it like a finished part.

6.7.2 Trimming

"Trimming" drop-down menu

Length of element:

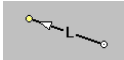
Change the length of a linear element. The starting point of the contour element remains unchanged.

- Closed contours: The manipulated element is recalculated – the position of the subsequent element is adapted.
- Open contours: The manipulated element is recalculated – the following contour train is shifted.

Operation

- ▶ Place the cursor on the contour element to be modified.
- ▶ Press the "confirm" soft key

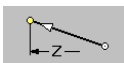
"Trimming" soft keys



New length



New end point



New end point

Continued ▶

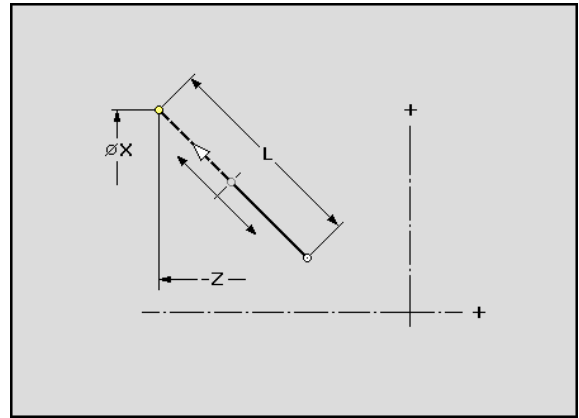
- ▶ Enter a new length/end position ("Change line length" dialog box)
- ▶ TURN PLUS depicts the changed contour
 - "Confirm" soft key accepts the solution
 - ESC key rejects the solution

Parameters

L/X/Z: ■ New length
 ■ New end position

Successor element:

- Modify angle to the next element ("with angle change")
- Do not modify angle to the next element ("without angle change")



Length of contour

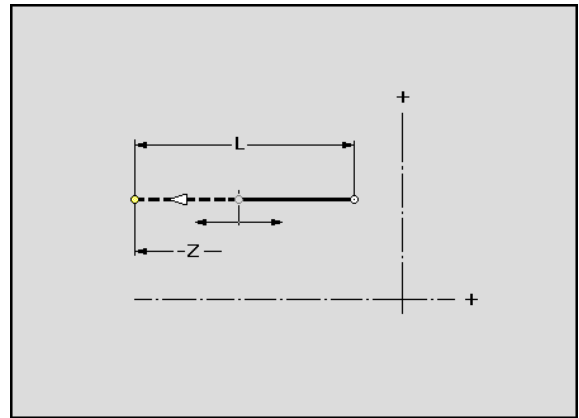
Change length of contour. Select the element to be modified and a "compensation element." These are usually one element of the outside contour and one of the inside contour.

Operation

- ▶ Place the cursor on the contour element to be modified.
- ▶ Press the "confirm" soft key
- ▶ Enter a new length or end position ("Change line length" dialog box)
- ▶ TURN PLUS depicts the changed contour
 - "Confirm" soft key accepts the solution
 - ESC key rejects the solution

Parameters

L/X/Z: ■ New length
 ■ New end position



Radius:

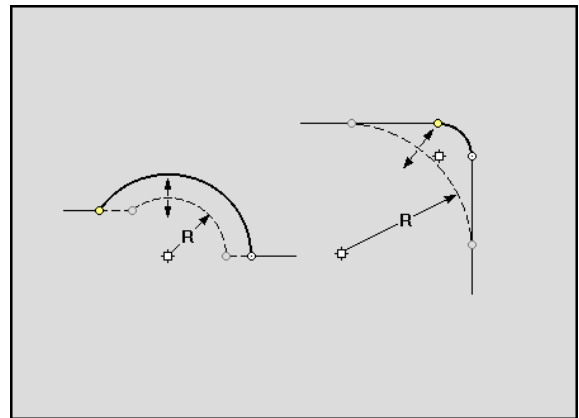
Change the radius of an arc.

Operation

- ▶ Place the cursor on the contour element to be modified.
- ▶ Press the "confirm" soft key
- ▶ Enter a new radius ("Trim radius" dialog box)
- ▶ TURN PLUS depicts the changed contour
 - "Confirm" soft key accepts the solution
 - ESC key rejects the solution

Parameters

R: Radius



Diameter

Change the diameter of a horizontal linear element. TURN PLUS recalculates the position of the manipulated element and adjusts the position of the preceding/succeeding element.

Operation

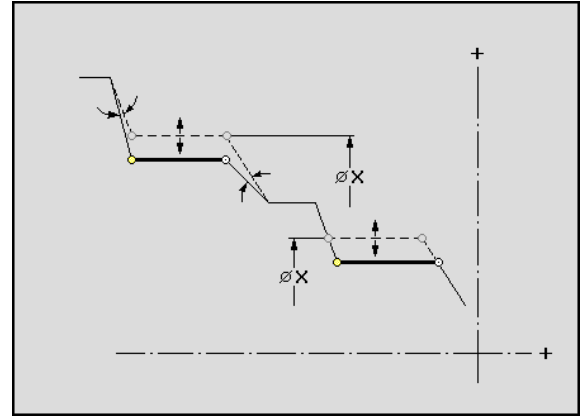
- ▶ Place the cursor on the contour element to be modified.
- ▶ Press the "confirm" soft key
- ▶ Enter the new diameter and adjustments to the predecessor/successor model ("Change diameter" dialog box)
- ▶ TURN PLUS depicts the changed contour
 - "Confirm" soft key accepts the solution
 - ESC key rejects the solution

Parameters of "Change diameter" dialog box:

X: New diameter

Preceding/succeeding element:

- "With angle change"
- "Without angle change"



6.7.3 Change

"Change" drop-down menu

Contour element

Change the parameters of the contour element. TURN PLUS adjusts the succeeding elements. The starting point stays as it is.

Operation

- ▶ Place the cursor on the contour element to be modified.
- ▶ Press the "confirm" soft key
- ▶ TURN PLUS opens a "Line/Arc" dialog box.
- ▶ Change parameters
- ▶ TURN PLUS depicts the changed contour
 - "Confirm" soft key accepts the solution
 - ESC key rejects the solution

"Contour element with shift"

Change the parameters of the contour element. TURN PLUS shifts the contour according to the new parameters. The starting point remains.

Operation

- ▶ Place the cursor on the contour element to be modified.
- ▶ Press the "confirm" soft key
- ▶ TURN PLUS opens a "Line/Arc" dialog box.
- ▶ Change parameters
- ▶ TURN PLUS depicts the changed contour
 - "Confirm" soft key accepts the solution
 - ESC key rejects the solution

Form element:

Change parameters of the form element. TURN PLUS adjusts the adjacent elements.

- ▶ Place the cursor on the form element to be modified.
- ▶ Press the "confirm" soft key
- ▶ TURN PLUS opens a dialog box containing the parameters of the form element.
- ▶ Change parameters
- ▶ TURN PLUS depicts the changed contour
 - "Confirm" soft key: Accepts the solution (When thread parameters are edited, the new parameters are immediately transferred.)
 - ESC key: Rejects the solution

"Pattern/Figure/Pocket":

Change parameters of the pattern/figure. If the contour consists of individual elements, you can enlarge or reduce (delete elements) it, or change its depth.

- ▶ Activate the window in the desired reference plane (front/back surface, Y front/Y back, Y surface).
- ▶ Place the cursor on the pattern/figure/contour.
- ▶ Press the "confirm" soft key

Pattern/figure: TURN PLUS opens a dialog box containing the parameters of the pattern/figure. – Change parameters

Extend the contour with a line segment/arc; Tag and delete a contour section with "Delete."
- ▶ TURN PLUS depicts the changed contour
 - "Confirm" soft key accepts the solution
 - ESC key rejects the solution

6.7.4 Deleting**"Delete" drop-down menu****Element/Range:**

Deletes the selected contour section

- Delete one **contour element**:
 - ▶ Place the cursor on the contour element.
 - ▶ "Confirm" soft key: TURN PLUS deletes the contour element
- **Delete contour range**:
 - ▶ Place the cursor on the beginning of the section.
 - ▶ Mark beginning of range ("range marking" soft key)
 - ▶ Place the cursor on the end of the contour section.
 - ▶ "Confirm" soft key: TURN PLUS deletes the section

Contour/Pocket/Figure/Pattern:

- Workpiece blank or finished part: Deletes the complete contour
- Pocket, figure, pattern:
 - ▶ Activate the window in the desired reference plane (front/back surface, Y front/Y back, Y surface).
 - ▶ Place the cursor on the pattern/figure/contour.
 - ▶ "Confirm" soft key: TURN PLUS deletes the contour element

Form element:

- ▶ Place the cursor on the form element.
- ▶ "Confirm" soft key: TURN PLUS deletes the form element and adjusts the reference element/ the adjacent elements.

All form elements:

TURN PLUS deletes all form elements and adjusts the reference elements/adjacent elements.

6.7.5 Inserting

"Insert" drop-down menu

"Line/Arc"

Inserts a linear element/an arc at the selected point.

- ▶ Select "Select point."
- ▶ OK soft key: TURN PLUS activates the "line menu/arc menu"
- ▶ Select and define the desired line/arc segment.
- ▶ TURN PLUS manipulates the contour accordingly.

"Contour"

Inserts several contour elements at the selected point.

- ▶ Select "Select point."
- ▶ "Confirm" soft key: TURN PLUS activates the element input
- ▶ Select and define the desired elements.
- ▶ TURN PLUS manipulates the contour accordingly.

6.7.6 Transformations

"Transformations" drop-down menu

The transformation functions are used for turning contours, contours on the end face or lateral surface, etc.

- Turning contour: The original contour is deleted and the complete turning contour is "transformed."
- Contours on the end face, lateral surface, etc.: Define whether you wish to delete the original contour or copy and "transform" it.

Shift:

Shifts the contour to the given position or incrementally (reference point: contour starting point).

Parameters

- X, Z: Target point
Xi, Zi: Target point – incremental

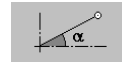
Turning:

TURN PLUS turns the contour about the **center of rotation** by the **angle of rotation**.

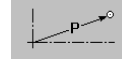
Parameters

- X, Z: Center of rotation in Cartesian coordinates
 α , P: Center of rotation in polar coordinates
W: Angle of rotation

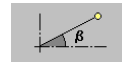
"Transformations" soft keys



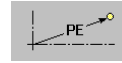
Polar dimensioning: Angle α



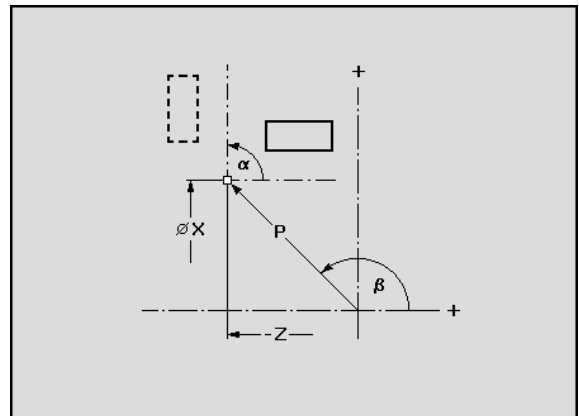
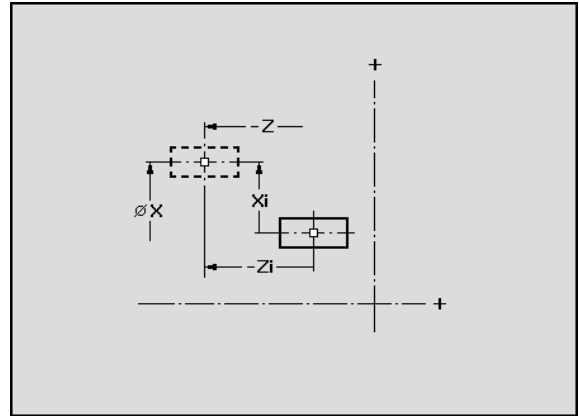
Polar dimensions: Radius



Polar dimensions end point: Angle β



Polar dimensions end point: Radius

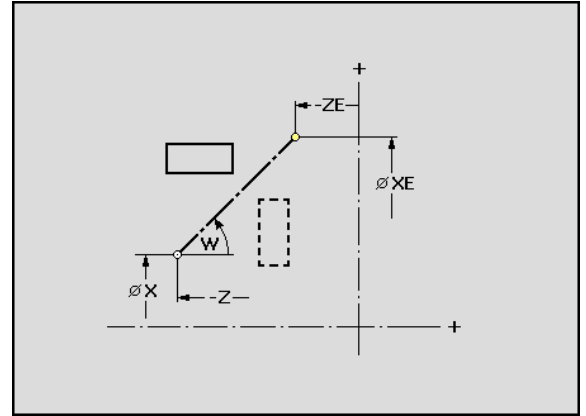


Mirroring:

You define the position of the **mirror axis** through the starting and end points or the starting point and angle.

Parameters

- X, Z: Starting point in Cartesian coordinates
- XE, ZE: End point in Cartesian coordinates
- W: Angle (reference: positive Z axis)
- α , P: Starting point in polar coordinates
- β , PE: End point in polar coordinates



"Invert":

Inverts the definition direction of the contour.

6.7.7 Connect

"Connect" menu item:

TURN PLUS closes an open contour by inserting a line segment.

6.7.8 Resolve

"Disconnect" menu item:

- Place the cursor on the form element/figure/pattern you wish to resolve.
- Press "confirm" soft key – TURN PLUS resolves the form element/figure/pattern
- Turning contour: Form elements (also chamfers and roundings) are transformed into linear segments and arcs.
- Contours on the end face or lateral surface, etc.: Figures and patterns are resolved into line segments and arcs.



Form elements/figures/patterns once resolved cannot be connected again.

6.8 Importing DXF Contours

6.8.1 Fundamentals

Contours available in DXF format can be imported into the TURN PLUS programming mode of operation.

DXF contours describe

- Workpiece blanks
- Finished parts
- Contour train
- Milling contours

For workpiece blank and finished-part contours as well as for contour trains, DXF layers should contain only one contour. For milling contours multiple contours can be contained and imported.

Requirements of a DXF contour or DXF file

- Only two-dimensional elements
- The contour must be in a separate layer (without dimension lines, without wraparound edges, etc.)
- Turning contours (workpiece blanks or finished parts) should be shown above the turning center (if this is not the case, then they must be revised in TURN PLUS)
- No complete circles, no splines, no DXF blocks (macros), etc.
- The imported contours can consist of up to 4000 elements (lines, arcs). In addition, up to 10 000 polyline points are permitted.

Preparation of contours during DXF import

The contour is converted from DXF format to TURN PLUS format during importation. The following changes are made to the contour representation, since there are basic differences between the DXF and TURN PLUS formats:

- Possible gaps between contour elements are closed
- Polylines are transformed into linear elements

In addition, the following features, which are necessary for a TURN PLUS contour, are specified:

- The starting point of the contour
- The rotational direction of the contour

Sequence of a DXF importation:

- ▶ Selection of the DXF file
- ▶ Selection of the layer, which contains only the contour(s)
- ▶ Import of the contour(s)
- ▶ Saving (and editing) of the contour in TURN PLUS

6.8.2 Configuring the DXF Import

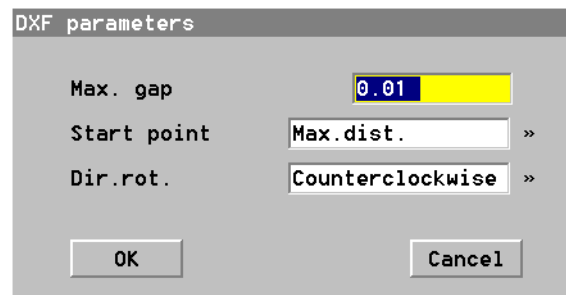
You can influence the preparation of the contour during the DXF importation with the configuration parameters described below.

DXF configuration:

- ▶ Starting from the main menu, select **Configuration/Change/DXF Parameters**
- ▶ Define the settings in the “DXF Parameters” dialog box
- ▶ Conclude with **OK**
- ▶ Call the “Settings” dialog box (**Settings** menu) and choose an entry for the **Start point automatic** field
- ▶ Conclude with **OK**
- ▶ Go back one menu step with the **ESC key**
- ▶ Select the **Configuration/Save** menu item
- ▶ Select the “Standard” file and save the changed configuration

DXF configuration parameters

- **Maximum gap:** There might be small gaps between contour elements in the DXF drawing. With this parameter you specify how large the distance between two contour elements may be.
 - If the **maximum gap** is not exceeded, then the following element is seen as being part of the “current” contour.
 - If the **maximum gap** is exceeded, then the following element is considered an element of the “new” contour.
- **Starting point:** The DXF import analyzes the contour and determines the starting point. The possible settings have the following meaning:
 - **Right, left, top, bottom:** The starting point is set to be the contour point that is the furthest to the right (or left, or ...). If more than one contour point satisfies this requirement, then one of these points is selected automatically.
 - **Maximum distance:** The DXF import sets the starting point to one of the two contour points farthest apart from each other. The program automatically determines which of these points is the starting point. It is not possible to influence this decision.
 - **Marked point:** If one of the contour points in the DXF drawing is marked with a complete circle, then this point is specified as the starting point. The contour point must be at the center of the complete circle.
- **Direction of rotation:** Indicate whether the contour is aligned in clockwise or counterclockwise direction.



In the **starting point automatic** configuration parameter you specify the behavior of TURN PLUS when entering the finished part contour.

Meaning of the setting in the **Starting point automatic** field:

- **Yes:** Upon calling the finished part contour entry, TURN PLUS immediately branches to the entry of the contour starting point. The **DXF import** soft key is not available.
- **No:** After the finished-part contour entry is called, you have the choice of whether a finished-part contour/DXF contour should be imported, or whether the contour will be entered manually.

Only the entry of the finished part contour is affected by this setting. For all other contours you select the form of contour entry either by menu or by soft key.

The screenshot shows a 'settings' dialog box with the following options:

Setting	Value	Action
Zoom behavior	dynamic	»
Level ID	Display	»
Point grid	Do not display	»
X value input	Diameter	»
With help graphic	Yes	»
Start point autom	Yes	»

At the bottom of the dialog are two buttons: 'OK' and 'Cancel'.

6.8.3 DXF-Import

The **DXF import** function is always offered when a contour entry is necessary. The sequence of a DXF import is independent of the type of contour to be imported (workpiece blank, finished part, etc.).

DXF import


Press the **DXF import** soft key. TURN PLUS opens the "DXF Import" selection box.

Choose and load a DXF file

Next contour

Select the contour to be imported with the **Next Contour** and **Previous Contour** soft keys

Previous contour



Import the DXF contour(s)

6.8.4 Transferring and Organizing DXF Files

The transfer and organization functions of the **Transfer** operating mode support DXF files.

In the "Mask of Files" dialog box, set the **TURN PLUS-DXF file** file type in order to edit DXF files.

6.9 Assigning attributes

Attributes for Workpiece Blanks

These attributes influence the division into machining areas and the selection of roughing cycles in the AWG.

Selection: "Workpiece – Blank – Attribute"

Finished part attributes

After the contour of a finished part has been defined, individual contour elements/contour sections can be assigned attributes. The AWG and IWG evaluate the attributes for generating a working plan.

Selection: "Workpiece – Finished part -Attribute"

6.9.1 Attributes for Workpiece Blanks

Define the "type semifinished" ("Surface quality" dialog box):

- **Cast or forged workpiece blank:** Working plan generation according to the "cast machining" strategy (first transversal – then longitudinal roughing).
- **Pretumed workpiece blank:** Working plan generation after the standard strategy. In addition to standard machining cycles, contour-parallel roughing cycles are used.
- **"Unknown" (or no attribute defined):** Working plan generation according to the standard strategy.

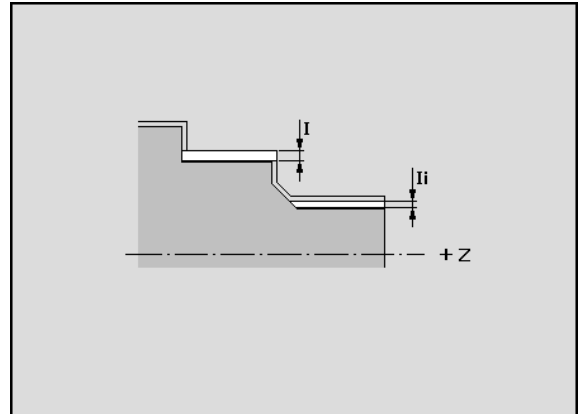
6.9.2 Oversize

The oversize, e.g. grinding allowance, is retained after the machining process has been completed. TURN PLUS distinguishes between:

- **Absolute oversize:** Is "final" – other oversizes are ignored.
- **Relative oversize:** Applies additively to other oversizes.

Parameters

I: Absolute oversize
li: Relative oversize



6.9.3 Feed rate/peak-to-valley height

Feed rate

The input value applies as finishing feed rate (see also "4.5.4 Auxiliary Commands of the Contour Description").

Feed rate reduction factor

The input value is multiplied by the current feed rate.

Peak-to-valley height

The peak-to-valley height is evaluated during finishing (see also "4.5.4 Auxiliary Commands of the Contour Description"). TURN PLUS distinguishes:

- Peak-to-valley height (Rt) – general peak-to-valley depth (profile depth)
- Mean roughness index (Ra)
- Average roughness value (Rz)

Additive correction

The CNC PILOT manages 16 tool-independent compensation values. In the dialog box, you enter the number of the additive correction value. The correction value is defined during the machining of the workpiece.

Exclusion from machining

The effect of the attribute depends on the type of machining:

- **Roughing:** The attribute is evaluated with the first/last element of an inside/outside contour. Form elements are not machined.
- **Finishing:** Marked elements are not finished.
- **Predrilling:** Attribute is ignored.
- **Recessing:** Marked recesses are not machined.
- **Thread machining:** Marked thread elements are not finished and threads are not cut.
- **Centering:** Marked holes (form elements) are not drilled.
- **Drilling:** Marked holes (for C/Y machining) are not machined.
- **Milling:** Marked milling contours (for C/Y machining) are not machined

6.9.4 Precision stop

Tagged contour elements are machined with the "precision stop" (see also "4.5.4 Auxiliary Commands of Contour Description").

6.9.5 Separation Points

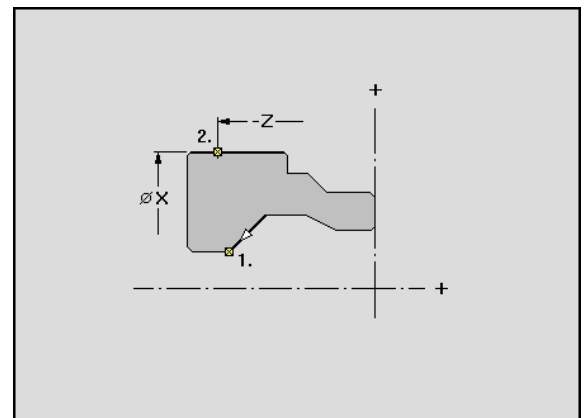
are used for shaft machining or machining with multiple setups. After the element is selected, TURN PLUS opens the "selection point" dialog box.

Parameters

Position:

- Delete: Deletes an existing separation point (a division of the contour element remains, however)
- 1. In target point: Separation point at the end of the element
- 2. On element: Separation point lies on the element

X, Z: Position of the separation point



6.9.6 Machining Attributes

The AWG evaluates the machining attributes for generating the working plan. The IWG uses the machining attributes as cycle parameters.

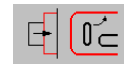
Defining machining attributes

- ▶ Adjust the working plane (turning contour, front face or lateral surface, etc.)
- ▶ Select the attribute type ("Machining attributes" drop-down menu)
- ▶ Select the contour element (existing attributes are displayed)
- ▶ Enter/edit the attributes

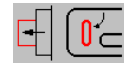
Soft keys

If holes or patterns and created in a figure (a figure in a figure), TURN PLUS differentiates between these "levels." First select the plane and the desired contour.

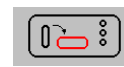
"Plane selection" soft keys



Next/previous plane in "figure in figure"



Next/previous plane in "figure in figure"



Next/previous figure or pattern

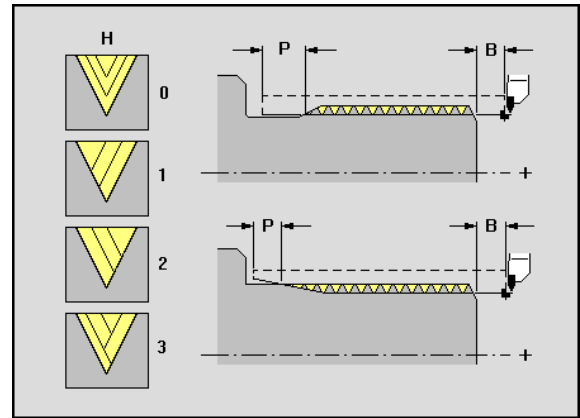


Next/previous figure or pattern

Machining attributes -Threading

Parameters

- B, P: Starting length, overrun length - no input: The CNC PILOT automatically determines the length from adjacent undercuts or recesses. If no undercut/recess exists, the starting length and overrun length from machining parameter 7 will be used (see also "4.8 Thread Cycles").
- C: Starting angle – if the beginning of the angle is defined with respect to rotationally nonsymmetrical contour elements
- I: Maximum approach - maximum infeed distance
- V: Type of infeed
- (V=0) Constant cross section: Constant cross section for all cuts
 - (V=1) Constant infeed
 - (V=2) (Remaining) cut division: If the division of thread depth by infeed yields a remainder, this remainder applies to the first infeed. The "last cut" is divided into 1/2, 1/4, 1/8 and 1/8 of a cut.
 - (V=3) EPL method: The infeed is calculated from the pitch and the speed
- H: Type of tool offset of the individual infeeds to smooth the thread flanks
- H=0: No offset
 - H=1: Offset from left
 - H=2: Offset from right
 - H=3: Offsets alternates right/left
- Q: Number of air cuts – after the last cut (for reducing the cutting pressure in the thread base)

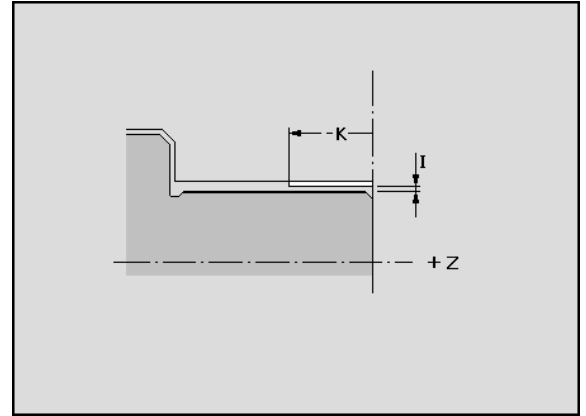


Machining attributes - Measuring

TURN PLUS uses the parameters of the "Measuring cut" dialog box to call the **expert program** in machining parameter 21 - "UP-MEAS01."

Parameters

- I: oversize for measuring cut
 K: Length of measuring cut
 Q: Measuring loop counter: Every nth workpiece is measured



Machining attributes - Drilling

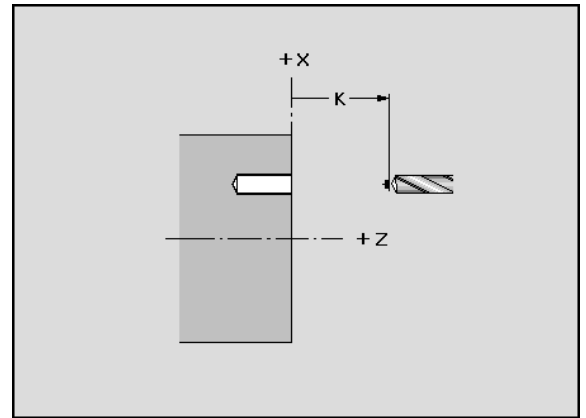
This attribute calls a submenu with drilling attributes and drilling combinations (see "4.9 Drilling Cycles"). TURN PLUS accounts for drilling combinations when selecting tools and generating the working plan (**one** machining cycle per "drilling combination").

Retraction plane

The tool is positioned to the return plane before/after each drilling operation (hole on lateral surface: diameter).

Parameters

- K: Retraction plane – Position of the drill before/after machining



Drilling combinations

The attribute influences the tool selection:

- **Centering and countersinking:** NC centering drill (type 32*);
alternative tool: centering drill (type 31*)
- **Counterboring:** Step drill (type 42*)
- **Tapping:** Tap (type 44*)
- **Drilling and reaming:** Delta drill (type 47*)

No machining

The hole/hole pattern is not machined.

Delete drilling attributes

Deletes all attributes of this hole.

Machining attributes - Milling

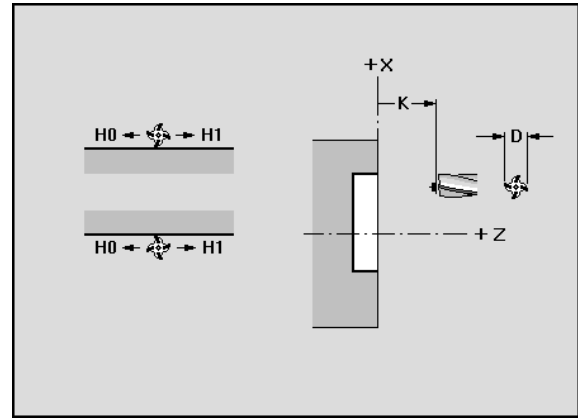
Select the desired type of milling operation from the drop-down menu (see also "4.11 Milling cycles").

Contour milling

Mills the figure or the "freely defined" open or closed contour.

Parameters

- Q: Milling location
- Contour: Center of cutter on the contour
 - Internal (milling) – closed contour
 - External (milling) – closed contour
 - To the left of the open contour (in machining direction)
 - To the right of the open contour (in machining direction)
- H: Milling direction
- 0: Up-cut mill
 - 1: Climb mill
- D: Milling diameter for selecting suitable tool.
- K: Retraction plane: Cutter position before/after milling (lateral surface: diameter).

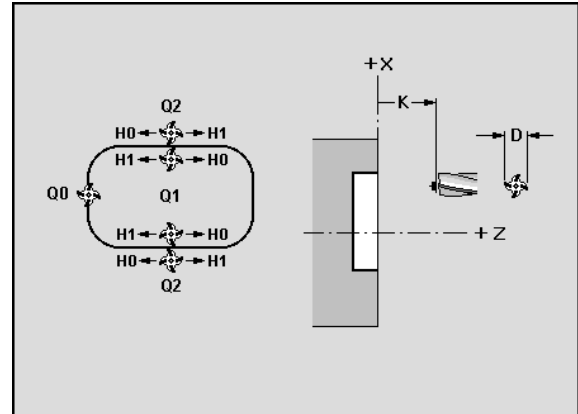


Area milling

Mills the internal surface area of closed contours (figure or "freely defined" contour).

Parameters

- H: Milling direction
- 0: Up-cut mill
 - 1: Climb mill
- D: Milling diameter for selecting suitable tool.
- K: Retraction plane: Cutter position before/after milling (lateral surface: diameter).

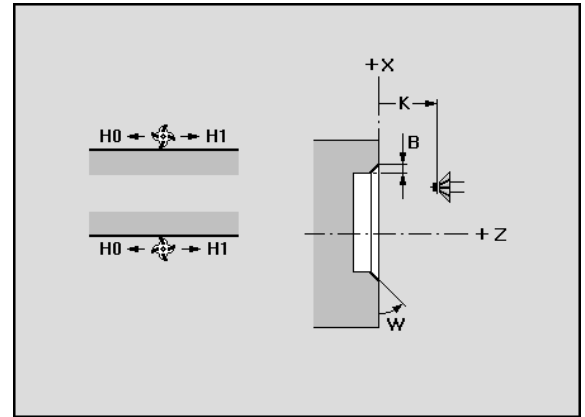


Deburring

Deburs the figure or the "freely defined" open or closed contour.

Parameters

- H: Milling direction
 ■ 0: Up-cut mill
 ■ 1: Climb mill
- B: Chamfer width
- W: Chamfer angle: For tool selection – default 45°
- K: Retraction plane: Cutter position before/after milling (lateral surface: diameter).

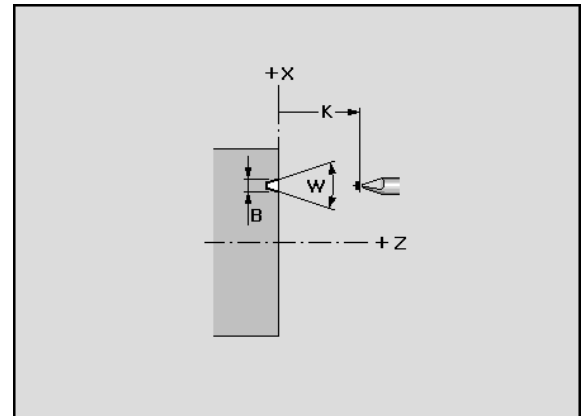


Engraving

Engraves a contour (figure, "freely defined" open or closed contours).

Parameters

- B: Width
- W: Angle for the tool selection – default 45°
- K: Retraction plane: Cutter position before/after milling (lateral surface: diameter).



Do not machine

The milling contour is not machined.

Delete milling attributes


Deletes all attributes of this milling contour.

6.10 User Aids

6.10.1 Calculator

You can use an online pocket calculator for example for standard calculations, calculation of fit tolerances, and calculation of the core hole diameter (for inside diameters).

To calculate:

- Position the cursor on the input field of the dialog box
-  Call the pocket calculator – the value of the input box is loaded.
- Perform the calculation.
- **"OK"** – deactivates the pocket calculator and accepts the value
- **"Delete"** - deactivates the calculator and rejects the value

Displays:

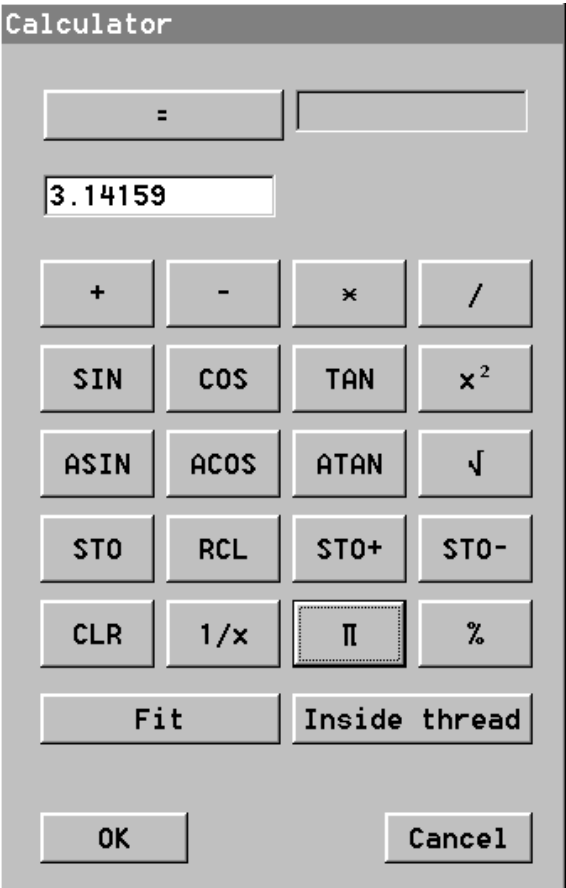
- Display value (below "=")
- saved value (right of "=")
- Operation and intermediate result (to the right of the display value)

Operation:

- Select and activate the function/box by cursor key or mouse.
- The arithmetical functions (SIN, squares, etc.) refer to the value displayed.

Calculator functions

=	Perform calculation; display result
+, -, *, /	Basic types of calculation
SIN, COS, TAN	Trigonometric functions
ASIN, ACOS, ATAN	Trigonometric inverted functions
X ²	Square
√	Square root
STO	Store value displayed
STO+, STO-	Add/subtract value displayed to memory contents
RCL	Recall memory contents to the display
CLR	Clear the display
1/x	Calculate the reciprocal of the value displayed
π	Ratio of circle circumference to diameter (3.14159)
n%	Percent calculation

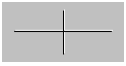


Calculator functions

- Fit
 - Calculate the mean tolerance for fits
 - Enter nominal diameter
 - Press "fit"
 - Enter the fit data ("fit" dialog box) – press "OK"
 - The calculator displays the mean tolerance value.
- To calculate a core hole diameter from thread data, use the "Internal thread" button.
 - Press "inside thread"
 - Enter the thread data ("inside thread" dialog box) – press "OK"
 - The calculator calculates the core hole diameter and accepts it as display value.

6.10.2 Digitizing

Input values can be determined and transferred to the input field using the quadruple arrow (digitizing). TURN PLUS displays the coordinates of the cross-hairs position.



Activate digitizing mode (with opened dialog box)

Position the cross hairs: Cursor keys or mouse

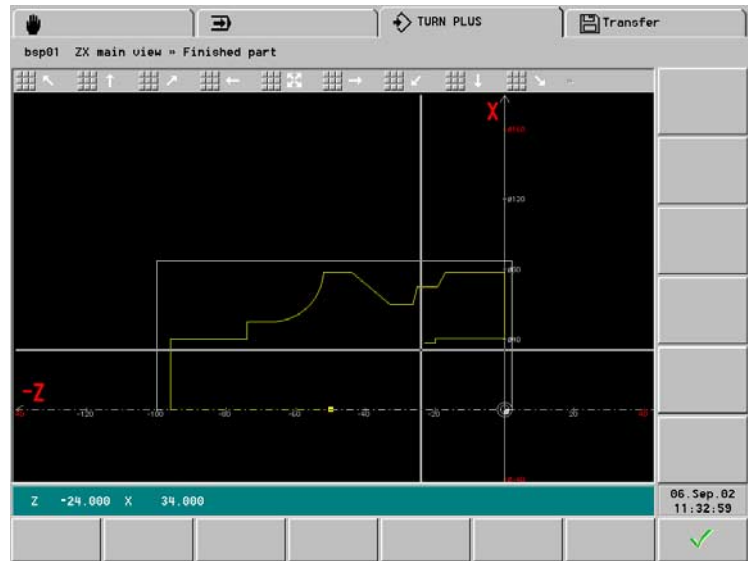
Exit digitizing mode:

- "Enter" – with value transfer
- "ESC" – without value transfer



Change the zoom setting **before** calling the digitizing mode if the increments of the cross hairs movements are too small/ large.

The values are taken over as **absolute values** of the Cartesian coordinate system – independent from the setting of the input fields.



6.10.3 Inspector – Checking Contour Elements

The "inspector function" can be used to check contour elements, form elements, figures and patterns. It is not possible to edit the displayed data.

Checking contour elements with the inspector function:

- Select the desired window (reference plane).



Inspector


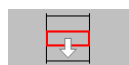

Call the "inspector"

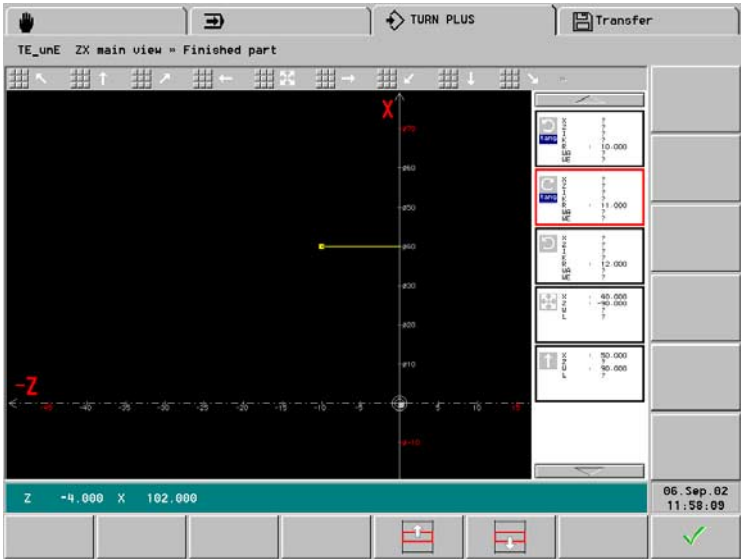
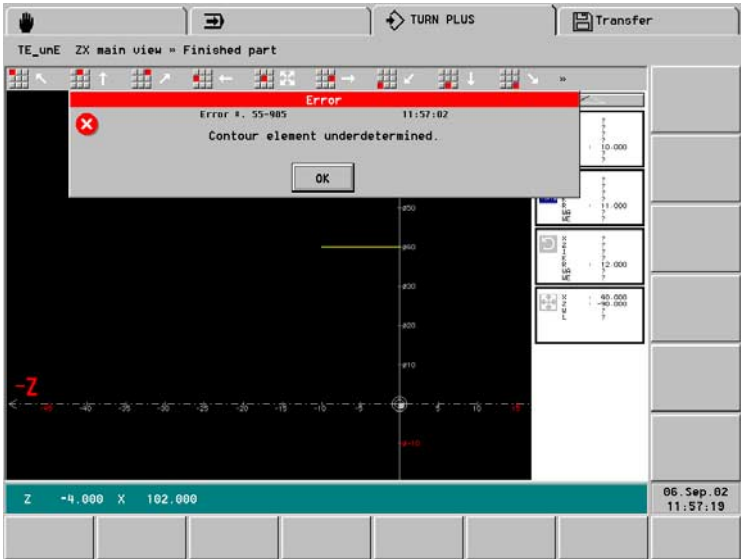
- Position the cursor on the contour/form element, figure or pattern and confirm
- TURN PLUS shows the **entered** parameters
 - ALT key: TURN PLUS shows **all** parameters of the element – for form elements the parameters of the individual elements
 - Arrow left/right (with opened dialog box): Shows the parameters of the following/previous element
- To close the dialog box, press the ESC key.

6.10.4 Unresolved Contour Elements

If a contour element is not sufficiently defined,TURN PLUS reports the error. After you have confirmed the error message, use the soft keys to position the cursor to the desired unresolved element and correct the data.

“Unresolved contour elements” soft keys

-  Mark previous unresolved element
-  Mark next unresolved element
-  Select marked unresolved element

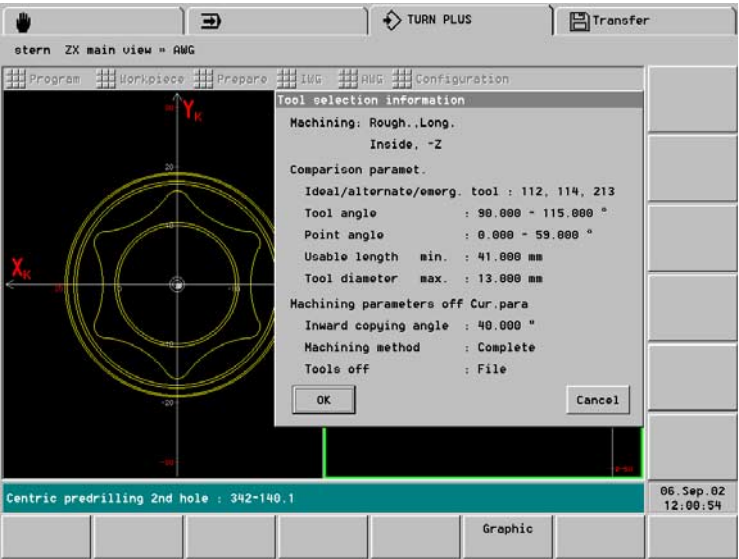
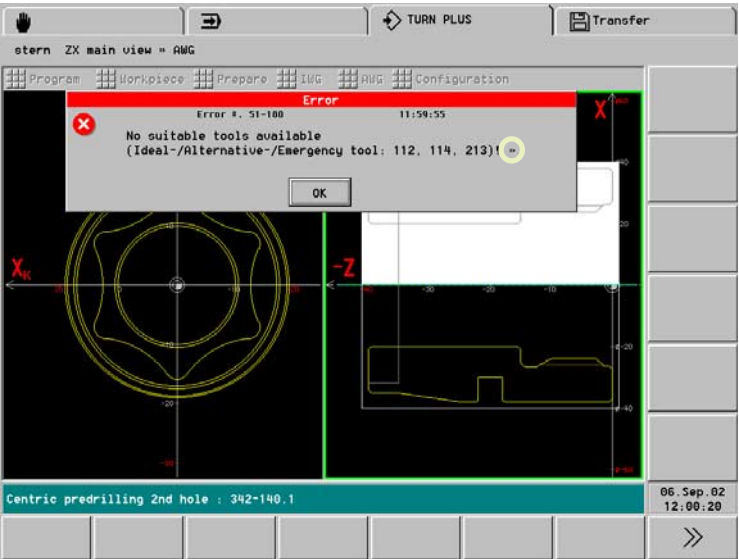


6.10.5 Error Messages

If the actual error message shows the characters ">>",TURN PLUS can display further information on the error, if desired.



Call the additional information on the error message



6.11 Preparing a Machining Process

The "Prepare" function defines the chucking equipment, the position of the chucking equipment, and TURN PLUS specific turret assignments.

For chucking a workpiece, TURN PLUS determines the

- Cutting limitation on the inside and outside of the contour
- Zero point shift (is taken into the NC program as a G59 command)

and transfers the following setup information to the program head (see "6.2.2 Program Head"):

- Clamping diameter
- Unclamping length
- Clamping pressure



- You can set/change the cutting limits.
- If you do not use "clamp" TURN PLUS assumed standard values.
- You define the chucking equipment for the second setup **after** machining the first setup.
- When the workpiece is clamped at the spindle and the tailstock, TURN PLUS assumes **shaft machining** (see also "6.16.9 Shaft Machining").

6.11.1 Chucking a Workpiece

Chucking a workpiece at the spindle

Chucking a workpiece at the spindle

Select "Prepare - Chucking - Chuck."

Select "Spindle side."

Select the type of chuck from the drop-down menu -TURN PLUS opens the corresponding dialog box:

- Two-jaw chuck
- Three-jaw chuck
- Four-jaw chuck
- Collet chuck
- Without chuck (face driver)
- Three-jaw chuck indirect (face driver in chuck with jaws)

- Enter the data for clamping
- Define the clamp range

TURN PLUS displays the selected chucking equipment and the cutting limit (red line).

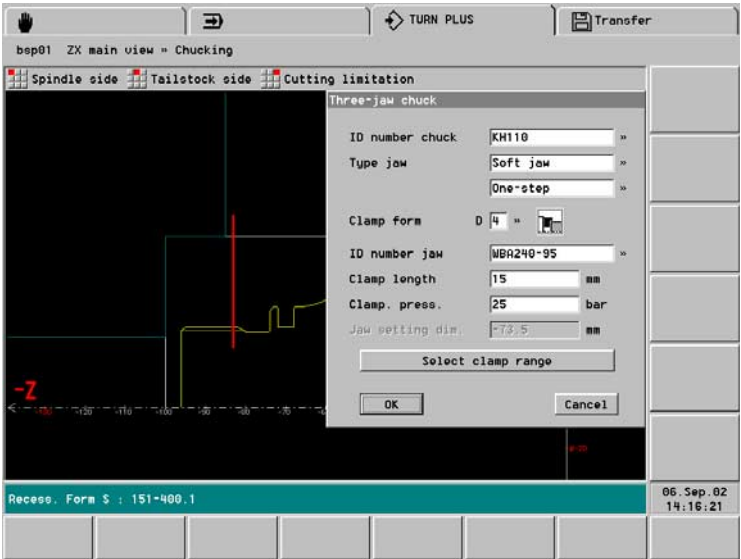


Select first the type of chuck and the jaw type. TURN PLUS takes these data into account in the selection of the ID number of the chuck jaw.

Continued ►

Parameters for two-jaw, three-jaw or four-jaw chucks:

- Chuck identification number
- Chuck type: Define the chuck type and steps
- Clamp form: Define the inside/outside clamp and clamp step
- Jaw identification number
- Clamping length: Ascertained from the jaw and the clamp form. Correct this value if the clamping length is different.
- Clamping pressure: is transferred to the PROGRAM HEAD –TURN PLUS does not evaluate the parameter.
- Jaw adjusting dimension: Distance between outer edge of chuck and outer edge of jaw.
Negative value: The jaw projects from the chuck (the dimension is for your information).
- “Select clamping range” button: Defines where to position the chuck
 - In the case of contours with chamfers, roundings or arcs, tag the section around the corner to be clamped.
 - In the case of rectangular contours, tag an element adjacent to the corner to be clamped.



Clamping form	Unstepped	One-step	Two-step
D=1			
D=2			
D=3			
D=4			
D=5			
D=6			
D=7			

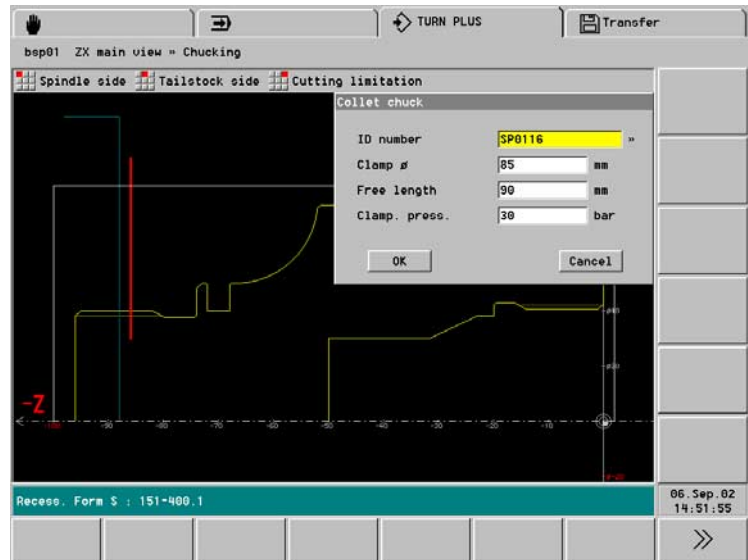
Collet chuck parameters:

Chuck identification number

Clamping diameter

Unclamping length: Distance between front edge of collet and right edge of blank

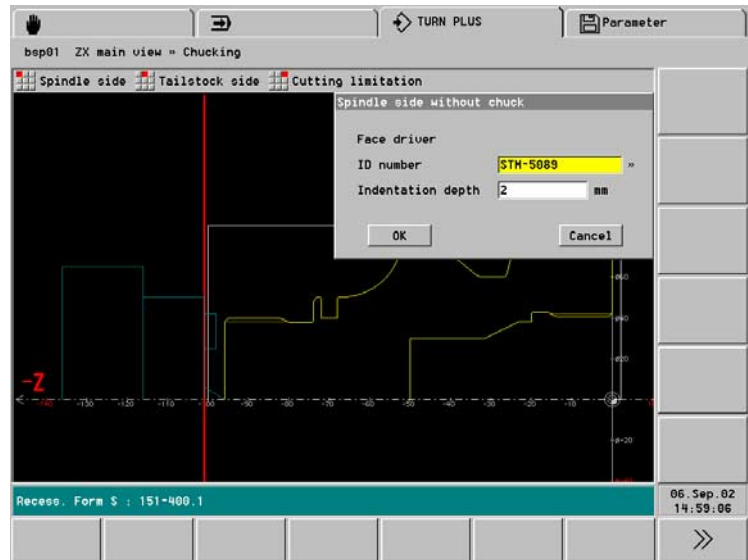
Clamping pressure: is transferred to the PROGRAM HEAD –TURN PLUS does not evaluate the parameter.



Parameters "without chuck" (face drive):

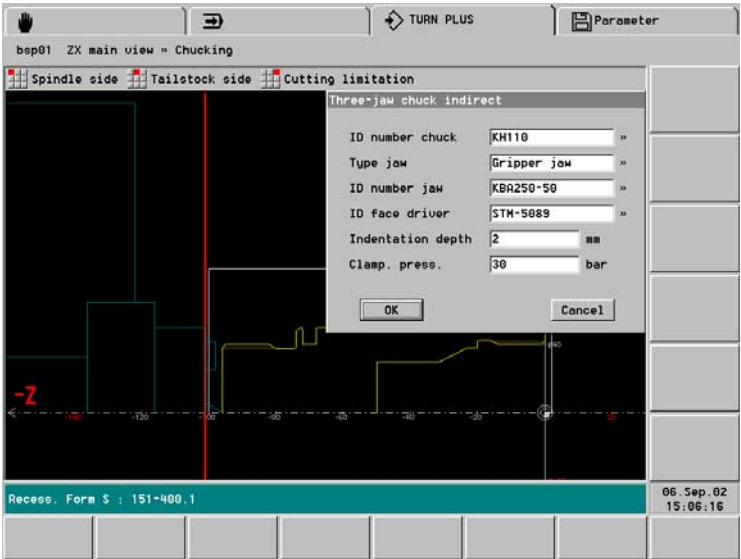
Identification number

Indentation depth: Approximate depth by which the claws indent the material (TURN PLUS uses this value to position the graphic representing the face driver).



Parameters for “Three-jaw chuck indirect” (face driver with jaws):

- Chuck identification number
- Jaw type: Select the jaw type.
- Jaw identification number
- Identification number of face driver
- Indentation depth: Approximate depth by which the claws indent the material (TURN PLUS uses this value to position the graphic representing the face driver).
- Clamping pressure: is transferred to the PROGRAM HEAD –TURN PLUS does not evaluate the parameter.



Chucking a workpiece at the tailstock

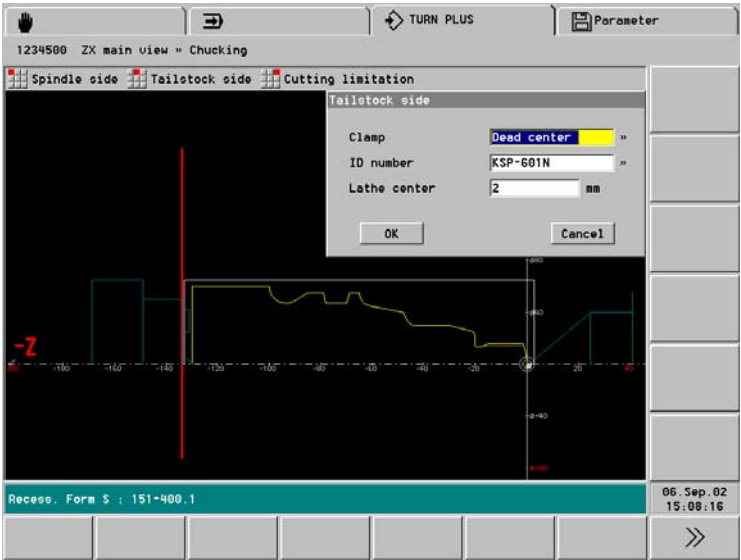
Menu item: Tailstock side

Parameters

- Chucking: Select the type of chucking equipment
 - Dead center
 - Lathe center
 - Centering taper
- Identification number of chucking equipment
- Centering depth: Depth by which the chucking equipment indents the material (TURN PLUS uses this value to position the graphic representing the chucking equipment).



If you clamp the workpiece at the spindle and the tailstock, TURN PLUS assumes that a **shaft is machined**.



Defining the cutting limit

Menu item: "Clamp – Cutting limitation"

TURN PLUS finds the "cutting limitation for AWG" on the outside and inside of the contour at "Clamp – Spindle side." You can edit or add to the values.

The cutting limit is displayed as a red line.

Deleting the chucking data

Menu item: "Chucking – Delete chuck plan"

Deletes all data for tool clamping and all entered cutting limitations.

Rechucking

Rechuck - Standard machining

Use "Rechuck - Standard machining" for front-face and rear-face machining with separate NC programs.

TURN PLUS

- "flips" the workpiece (blank and finished part) and shifts the zero point by "Nvz"
- Rotates lateral surface contour or contours of the YZ plane about "Wvc"
- Deletes the chucking equipment of the first setup.

"Rechuck workpiece" parameters

Nvz: Zero point shift (proposed value: length of the finished contour)

Wvc: Angular shift

"Rechuck – Complete machining 1st chucking after 2nd chucking"

Starts machining of the second setup.

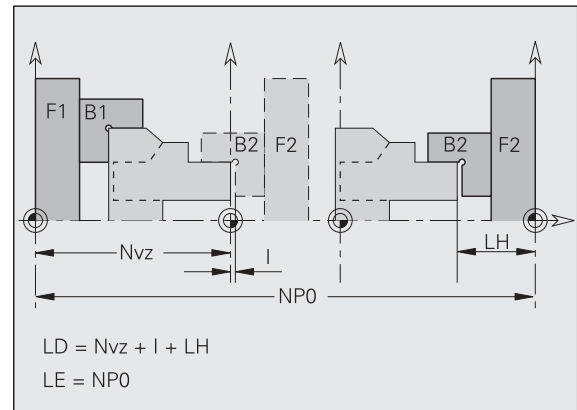
First define the chucking equipment. Then, TURN PLUS activates an **expert program** (from machining parameter 21) for workpiece transfer. Which expert program is used depends on the entry "1st setup of spindle .. – 2nd setup of spindle .." in the program header:

- Same spindle (manual rechucking): Enter "UP-UMHAND"
- Different spindles (transfer of workpiece to the opposing spindle): Enter "UP-UMKOMPL"

Expert programs are provided by the machine tool builder. That is why there may be deviations in the parameters described below. Use the expert program or the machine manual to inform yourself of the meaning of the parameters and the process of the expert program.



- Before rechucking, save the working plan, etc. for machining the first setup. When you use the rechucking function, TURN PLUS deletes the previous working plan and the operating resources used.
- Rechucking is no substitute for chucking.



- F1/B1, F2/B2: Chuck/Chuck jaw for main and opposing spindle
- Nvz: Zero point shift (G59, ...)
- I: Safety clearance on workpiece blank (machining parameter 2)
- NP0: Zero point offset (e.g. machine parameter 1164 for Z axis \$1)

Continued ►

TURN PLUS entered the calculated parameters as proposed values. Check and edit the entries.



The meaning of the transfer program depends on the name of the expert program.

Transfer parameters with the expert program "UMKOMPL"

Spindle speed for workpiece transfer (LA)

Direction of spindle rotation (LB):

- ☐ 0: CCW
- ☐ 1: CW

Speed or angular synchronism (LC):

- ☐ 0: Angular synchronism without angular offset
- ☐ >0: Angular synchronism with preset angular offset
- ☐ <0: Spindle speed synchronism

Transfer position in Z (LD):

- ☐ 0: Transfer position in machine dimension 1
- ☐ 1..6: Transfer position in machine dimension 1..6
- ☐ ≠ 0..6: Transfer position – Calculation of the proposed value: see sketch

Working position in Z (LE):

Proposed value: Zero point offset e.g. from machine parameter 1164 for Z axis \$1 (see sketch)

Finished part length (LF): From the workpiece description

Distance from stop surface (LH): Distance between reference point of chuck and stop surface of the clamping jaw, calculated from the second setup

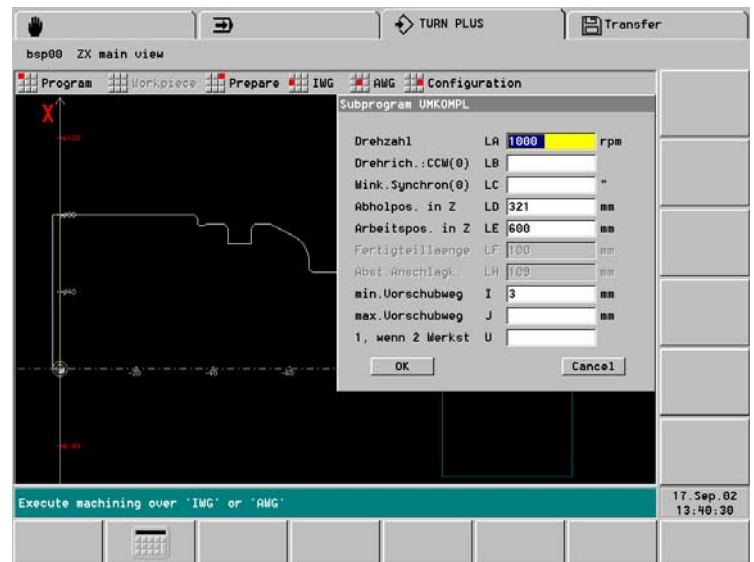
Minimum traverse (I):

- ☐ Without moving to fixed stop: Safety clearance on the workpiece to be transferred – proposed value: From "safety clearance on blank" (machining parameter 2)
- ☐ With traverse to a fixed stop: See machine manual

Maximum traverse (J):

- ☐ No input: Without traverse to fixed stop
- ☐ With input: With traverse to fixed stop – meaning of the parameters I and J: See machine manual

1, if 2 workpieces (U): No meaning



Continued ►

Transfer parameters in expert program with other name

Spindle speed for workpiece transfer (LA)

Direction of spindle rotation (LB):

- 3: CW
- 4: CCW

Angular synchronization (LC):

- 0: Angular synchronization
- 1: Speed synchronization

Offset angle (LD): with angular synchronization

Dead stop (LE):

- 0: With traverse to dead stop
- 1: Without traverse to dead stop

Transfer dimension (LF): Transfer position in machine dimension n (n: 1..6)

Minimum feed path (LH): For "traversing to dead stop" (see machining manual)

Maximum feed path (LH): For "traversing to dead stop" (see machining manual)

Feed path (J): For "traversing to dead stop" (see machining manual)

Jaw rinsing (K): see machine manual

Transfer parameters – for information

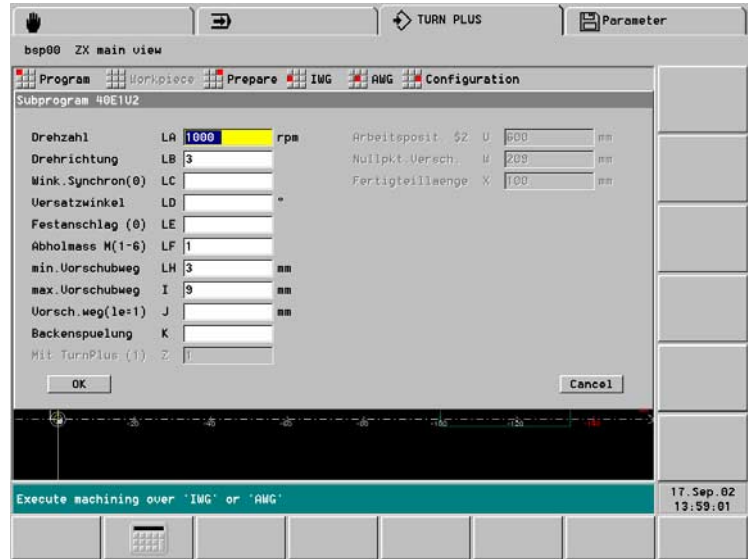
With TURN PLUS (Z):

- 1: Prepare work on the opposing spindle (switch-on conversions, zero point shift, etc.)

Working position \$2 Z (U): Proposed value: Zero point offset e.g. from machine parameter 1164 for Z axis \$1 (see sketch)

Zero point shift (W): Shift of the NC zero point (calculation: distance from reference point on chuck to dead stop on chuck jaw + finished part length)

Finished part length (LF): From the workpiece description



"Rechuck - Full-surface machining back to 1st chucking"

If you wish to correct or optimize the contour or machining process after the 2nd setup has been generated, you can use this function to return to the starting point of your machining process. The working blocks of the 2nd setup are rejected.

6.11.2 Setting Up a Tool List

With "Prepare –Tool list – ..." you manage **custom turret assignments** (see also "Machining Parameters 2 GlobalTechnology Parameters").

- **View turret – look at turret n:** Shows the turret assignment.
- **Set up turret – set up turret n:** Select tools and position them on the turret
- **Load list – saved tool list:** Load a saved tool list ("Load file" selection box)
- **Load list – tool list of the machine:** Accept the current revolver assignment of the machine (see "3.3.1 Setting Up aTool List").
- **Save list:** Save the current turret assignment in a file
- **Delete list:** TURN PLUS deletes the selected file



LoadTURN PLUS-specific turret assignments **before** using the IWG/AWG function for selecting tools.

Tools from the database

Select "Setting up -Tool list - Set up turret – Set up turret n"

Select the tool pocket (arrow up/down or touch pad)

Type list

Enter the tool type – the CNC PILOT displays all tools of this type mask

ID list

Enter the ID number – the CNC PILOT displays all the tools of this ID mask

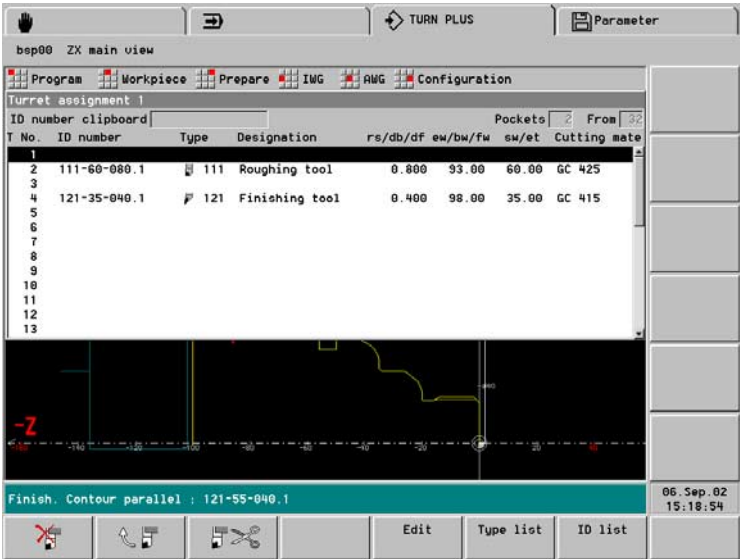
Select the tool

Insert




Take the tool from the database

Esc

ESC key – exit the tool database



"Tool database" soft keys

-  Delete tool
-  Take the tool from the "ID number clipboard"
-  Delete the tool and place in the "ID number clipboard"
- Edit

 Edit the tool parameters
- Type list

 Entries in the tool database - sorted by tool type
- ID list

 Entries in the tool database - sorted by tool ID number

For further soft keys see "3.3.1 Setting Up aTool List"



Adjust the coolant circuits in the "tool" dialog box.

Enter a new tool

Select "Setting up -Tool list - Set up turret – Set up turret n"

Select the tool pocket (arrow up/down or touch pad)


ENTER (or INS key) – opens the "tool" dialog box

- Enter the tool ID number
- Button **Coolant circuit**: Set the displays circuits (on, off, high pressure)


Changing the tool pocket

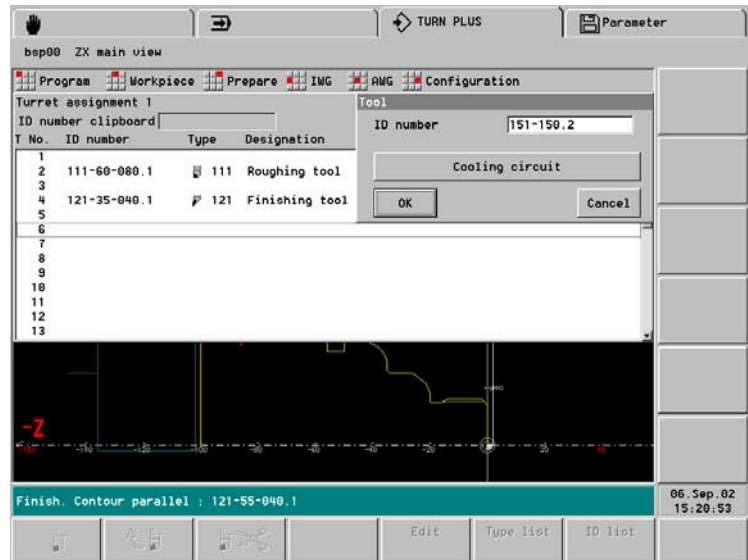
Select "Setting up -Tool list - Set up turret – Set up turret n"

Select the tool pocket (arrow up/down or touch pad)

 Deletes the tool and places it in the "ID number clipboard"

Select a new tool pocket (arrow up/down or touch pad)

 Take the tool from the "ID number clipboard"
If the location was occupied, the previous tool is taken into the clipboard.



Delete tool

Select "Setting up -Tool list - Set up turret – Set up turret n"

Select the tool pocket (arrow up/down or touch pad)

 or the DEL key deletes the tool

6.12 Interactive Working Plan Generation (IWG)

In the **IWG**, you define the individual **blocks** of the working plan. You do this by selecting the tool and the cutting values and determining the fixed cycle.

The **semiautomatic mode** generates a complete work block (part machining).

In **special machining (SM)** you add paths of traverse, subprogram calls or G/M functions (example: use of tool handling systems).

A working block may have the following content:

- The tool call
- The cutting data (technology data)
- Approaching the workpiece
- Fixed cycle
- Retraction
- Approaching the tool change position

When the tool data/cutting data of the previous working block are used, TURN PLUS does not generate a new tool call or commands for a new feed rate/spindle speed.

Generating a Working Block

Select the required machining operation.

Select the tool ("Tool" submenu)

Select "Cutting data"

- Check/optimize the cutting data.
- Activate/deactivate the coolant and define the coolant circuit

Select "Cycle - Machining range."

- Define the machining range by selection
- TURN PLUS marks the selected area

Select "Cycle - Cycle parameters"

- TURN PLUS opens the "cycle parameters" dialog box
- Check/optimize the parameters



Generating a Working Block (continued)

If required, select "Cycle - Approach."

- Enter approach position and type of approach

If required, select "Cycle - Retract tool."

- Enter the position and type of retraction

If required, select "Cycle - Move to tool change point."

- Enter position and type of approach to the change point

"Start" – TURN PLUS stimulates the machining (see "6.14 Control Graphics")

You can accept the work block:

- **Accept:** The work block is saved and the workpiece contour updated (the work block is saved and the workpiece is updated (regeneration of the blank))
- **Change:** TURN PLUS rejects the work block – check/optimize the parameters and simulate again
- **Repeat:** TURN PLUS simulates machining again

Continue existing working plan

Select "IWG"

TURN PLUS opens the "work plan exists" dialog box. Select **continue**

Add further work blocks

Change existing work plan

Select "IWG"

TURN PLUS opens the "Working plan exists!" dialog box. Select **Change**

TURN PLUS displays the existing work plan

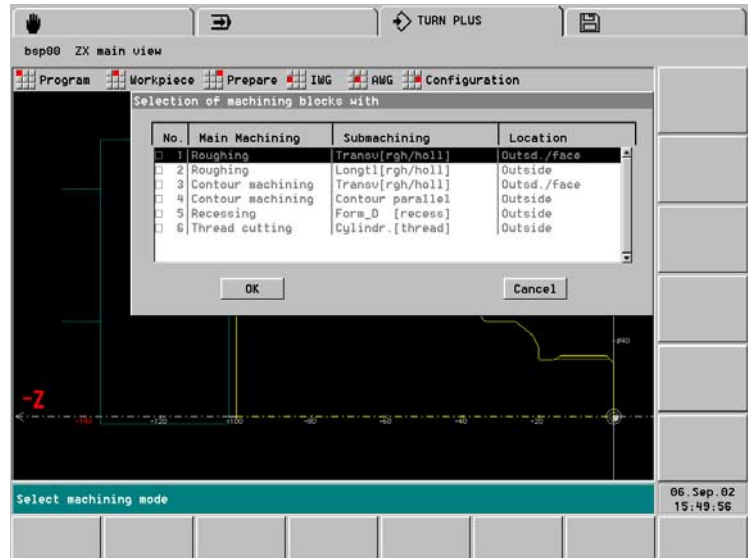
Mark the work blocks to be changed

TURN PLUS simulates the work plan

- Unmarked work blocks: Without stop
- Marked work blocks: Question "Change?"

Work blocks to be changed:

- TURN PLUS marks the machining range and makes all IWG functions available
- Correct/optimize the machining block



6.12.1 Tool call

"Tool – ..." drop-down menu

- **Manually through turret assignment:** Select a tool positioned on the turret
- **Manually through tool type/ID number:** Select the tool from the database and position it on the turret
- **From last machining process:** Select the tool used for the last machining operation.
- **Automatic:** The IWG selects and places the tool in the revolver. - Precondition: Definition of the machining range.

6.12.2 Cutting Data

- The cutting speed, main and auxiliary feed rate are determined from the material and the tool data. Check/optimize the values
- Maximum cutting depth P is adopted as cycle parameter.
- Define the coolant and coolant circuit

6.12.3 Cycle specification

Menu item "Cycle - ..."

Machining range: Use the range selection function to set the area to be machined.

Cycle parameter: Check/optimize the parameters.

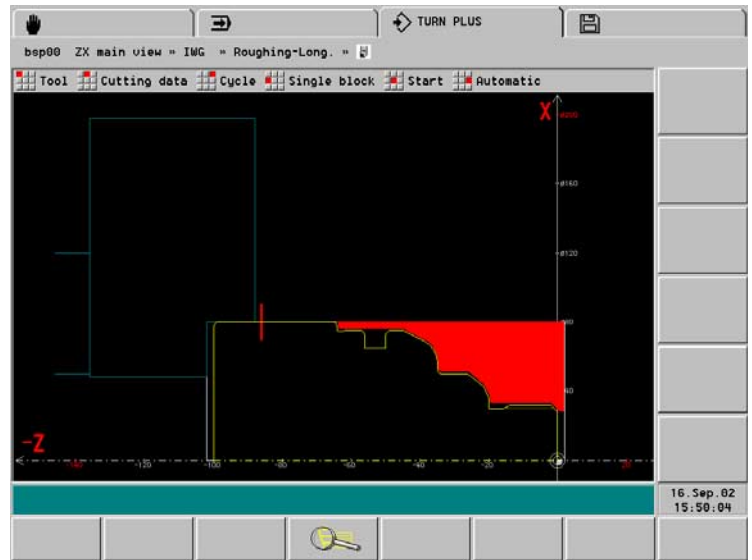
Approach: The tool moves at rapid traverse from the current position to the tool change position – before the cycle is called.

The "Approach" function is not contained in drilling and threading cycles. Place the tool on a suitable position using the "Approach" function.

Retract: After the cycle is finished, the tool moves at rapid traverse to the retraction position.

Move to tool change position: After the cycle is finished or after "retraction," the tool moves at rapid traverse to the change position. The change position defined in the dialog box is evaluated only when "WP=1" (machining parameter 2).

The traverse type (G0 or G14) and the tool change position are defined in machining parameter 2.



Machining direction during range selection:

- By key or soft key: The sequence of selection determines the machining direction
- Touch pad:
Left mouse key – machining direction is the direction of contour creation;
Right mouse key – machining direction is opposed to the direction of contour creation

6.12.4 Roughing

Overview of roughing operations

- Roughing longitudinal (G810)
- Roughing transverse (G820)
- Contour parallel roughing (G830)
- Roughing automatic –TURN PLUS generates **all** roughing operations automatically
- Rough hollowing
 - Residual roughing longitudinal
 - Residual roughing transverse
 - Residual roughing contour parallel
 - automatic hollowing
- Rough hollowing (neutral tool)

Roughing longitudinal, transverse (G810, G820)

Parameters

- P: Cutting depth (maximum infeed)
- A: Approach angle (reference: Z axis)
- Longitudinal: Default 0°/180° (parallel to Z axis)
 - Plan: default 90°/270° (perpendicular to Z axis)
- W: Departure angle (reference: Z axis)
- Longitudinal: Default 90°/270° (perpendicular to Z axis)
 - Transverse: Default 0°/180° (parallel to Z axis)
- X, Z: Cutting limit

Type of oversize is selected by soft key per Softkey

- I, K: Different longitudinal/transverse oversize
- I: Constant oversize – generates "oversize G58" before the cycle

Plunging: Machine descending contours ?

- Yes
- No

E: Reduced plunging feed rate with descending contours

- H: Type of departure – type of contour smoothing
- H=0: Smoothing after each cut along the contour
 - H=1: Lift off at under 45°; contour smoothing after the last cut
 - H=2: Lift off at under 45° – no contour smoothing

- Q: Retraction at cycle end
- Q=0: Return to starting point
 - Longitudinal: first X, then Z direction
 - Transverse: First Z, then X direction
 - Q=1: Positions in front of the finished contour
 - Q=2: Lifts off to safety clearance and stops

Undercutting (see soft-key table)

"Roughing" Soft key



Select longitudinal/transverse oversize or constant oversize.



FD relief turn machining



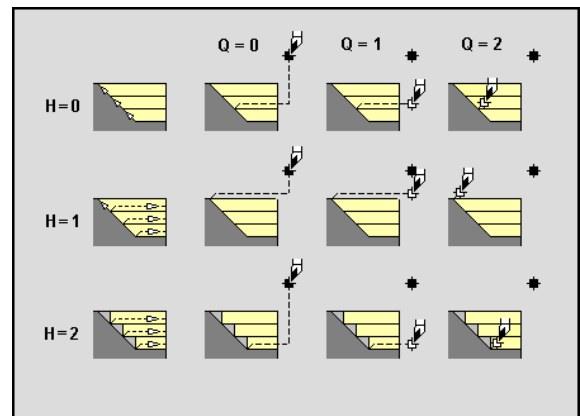
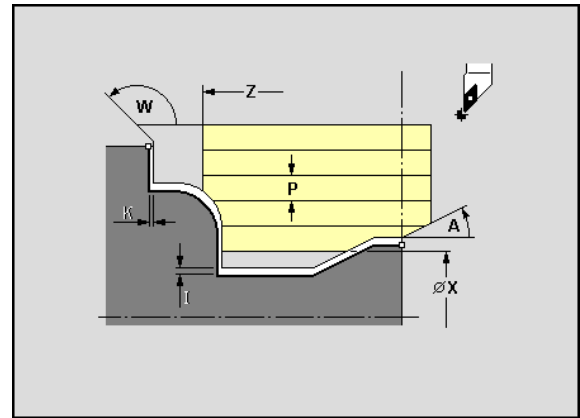
E and F undercut machining



G undercut machining



H, K and U undercut machining



Roughing contour-parallel (G830)

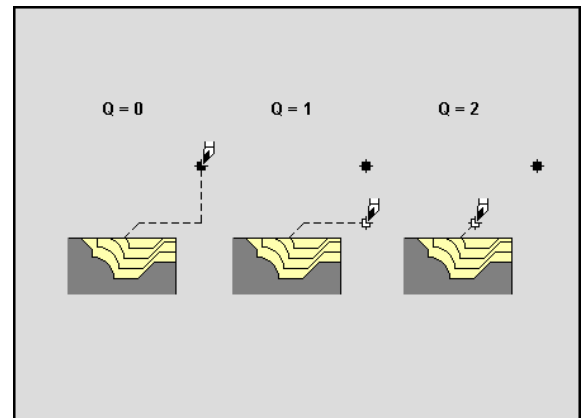
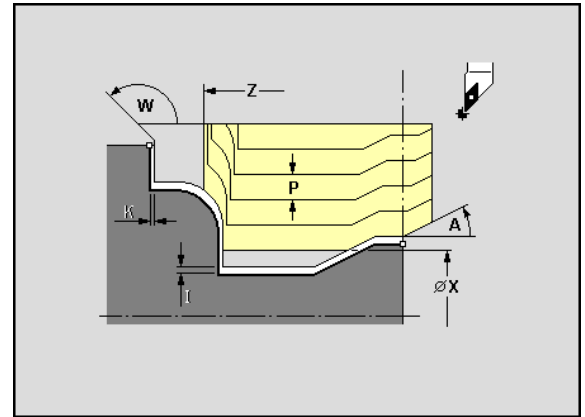
Parameters

- P: Cutting depth (maximum infeed)
A: Approach angle (reference: Z axis) – default 0°/180° (parallel to Z axis)
W: Departing angle (reference: Z-axis) – default 90°/270° (perpendicular to Z-axis)
X, Z: Cutting limit

Type of oversize is set by soft key

- I, K: Different longitudinal/transverse oversize
I: Constant oversize – generates "oversize G58" before the cycle
E: Reduced plunging feed rate
Q: Type of retraction at cycle end
■ Q=0: Return to starting point – first X, then Z direction
■ Q=1: Positions before the finished contour
■ Q=2: Lifts off to the safety clearance and stops

Undercutting (see soft-key table)



Roughing automatic

Selection: Roughing – Roughing automatic

TURN PLUS generates the working blocks for all roughing operations (longitudinal, transverse, hollowing, inside, outside, etc.). All the elements contained in a working block are determined (tools, cutting data, cycle parameters, etc.).

Cutting limitation with residual roughing

Using the "Roughing hollowing - residual roughing .." function, you can remove residual material from sloping contours.

Cutting limitation: If no cutting limits are defined, TURN PLUS machines in the selected area. To avoid collision, the select machining area is reduced by the cutting limitation function. The machining cycle accounts for the safety clearance (SAR, SIR – machining parameter 2) in front of the remaining material.

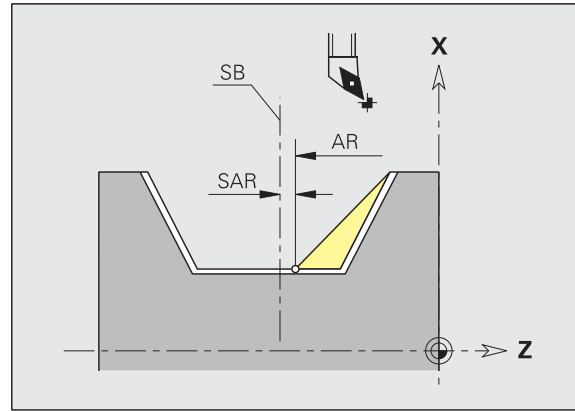
Defining the cutting limitation

- Position the tool so that it moves on the same side as the residual material.
- Selecting the machining range
- Select the starting point of the remaining material as the position of the **cutting limitation** (see figure).



Danger of collision

The residual material is machined without monitoring for collision. Check the cutting limits and the angle of approach given in the "Cycle parameter (roughing)" dialog box.



- AR: Starting point of residual material
- SAR: External safety clearance (machining parameter 2)
- SB: Cutting limit

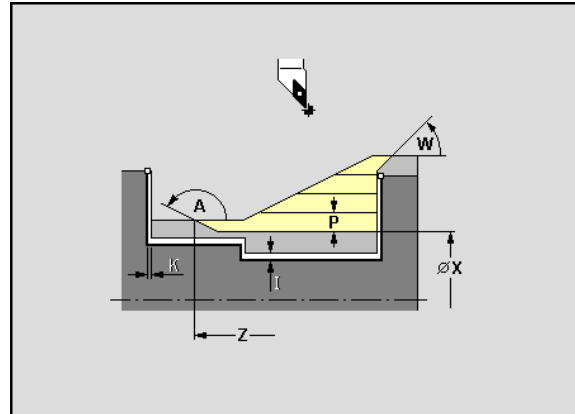
Residual roughing (hollowing) – longitudinal/transverse

Parameters

- P: Cutting depth (maximum infeed)
- A: Approach angle (reference: Z axis)
 - Longitudinal: Default 0°/180° (parallel to Z axis)
 - Plan: default 90°/270° (perpendicular to Z axis)
- W: Departure angle (reference: Z axis)
 - Longitudinal: Default 90°/270° (perpendicular to Z axis)
 - Transverse: Default 0°/180° (parallel to Z axis)
- X, Z: Cutting limit

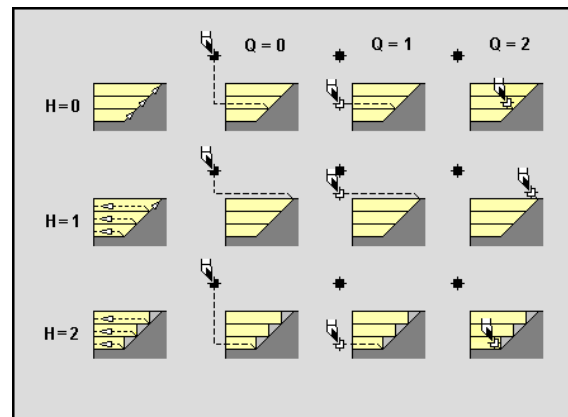
Type of oversize is selected by soft key per Softkey

- I, K: Different longitudinal/transverse oversize
- I: Constant oversize – generates "oversize G58" before the cycle
- Plunging: Machine descending contours?
 - Yes
 - No
- E: Reduced plunging feed rate with descending contours



Continued ►

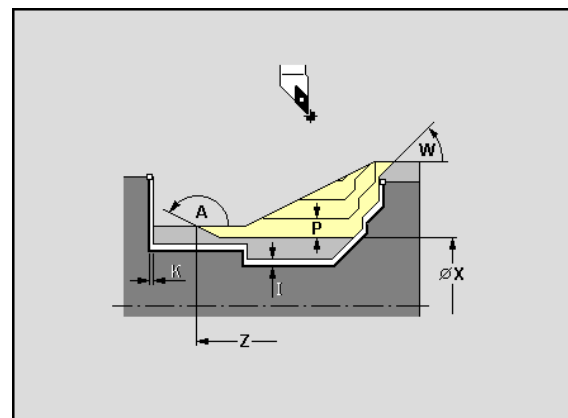
- H: Type of departure – type of contour smoothing
- H=0: Smoothing after each cut along the contour
 - H=1: Lift off at under 45°; contour smoothing after the last cut
 - H=2: Lift off at under 45° – no contour smoothing
- Q: Retraction at cycle end
- Q=0: Return to starting point
 - Longitudinal: first X, then Z direction
 - Transverse: First Z, then X direction
 - Q=1: Positions in front of the finished contour
 - Q=2: Lifts off to safety clearance and stops
- Undercutting (see soft-key table)



Residual roughing (hollowing) – contour parallel

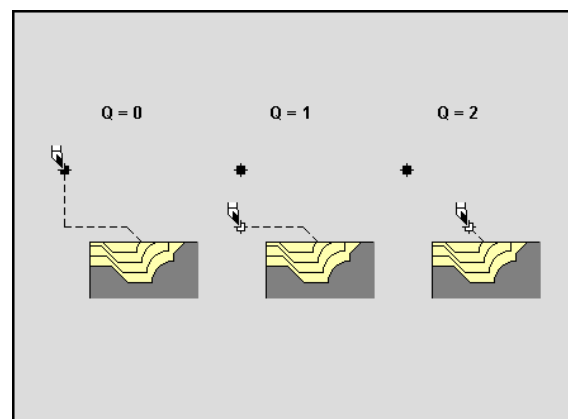
Parameters

- P: Cutting depth (maximum infeed)
- A: Approach angle (reference: Z axis)
- Longitudinal: Default 0°/180° (parallel to Z axis)
 - Plan: default 90°/270° (perpendicular to Z axis)
- W: Departure angle (reference: Z axis)
- Longitudinal: Default 90°/270° (perpendicular to Z axis)
 - Transverse: Default 0°/180° (parallel to Z axis)
- X, Z: Cutting limit
- Type of oversize** is selected by soft key per Softkey
- I, K: Different longitudinal/transverse oversize
- I: Constant oversize – generates "oversize G58" before the cycle



- Plunging: Machine descending contours ?
- Yes
 - No
- E: Reduced plunging feed rate with descending contours
- H: Type of departure – type of contour smoothing
- H=0: Smoothing after each cut along the contour
 - H=1: Lift off at under 45°; contour smoothing after the last cut
 - H=2: Lift off at under 45° – no contour smoothing
- Q: Retraction at cycle end
- Q=0: Return to starting point
 - Longitudinal: first X, then Z direction
 - Transverse: First Z, then X direction
 - Q=1: Positions in front of the finished contour
 - Q=2: Lifts off to safety clearance and stops

Undercutting (see soft-key table)



Hollowing –Automatic

supports **bilateral machining**. TURN PLUS first selects a roughing tool for rough-machining and then a tool for removing the residual material in the opposite machining direction.



You can use the "Hollowing automatic" function only for **recesses** (a relief turn can be machined using a standard roughing cycle). TURN PLUS uses the "Permissible inward copying angle EKW" (machining parameter 1) to distinguish recesses from relief turns.

Roughing hollowing – neutral tool (G835)

Parameters

- P: Cutting depth (maximum infeed)
A: Approach angle (reference: Z axis) – default 0°/180° (parallel to Z axis)
W: Departing angle (reference: Z-axis) – default 90°/270° (perpendicular to Z-axis)
X, Z: Cutting limit

Type of oversize is set by soft key

- I, K: Different longitudinal/transverse oversize
I: Constant oversize – generates "oversize G58" before the cycle

Plunging: Machine descending contours ?

- ☐ Yes
☐ No

E: Reduced plunging feed rate with descending contours

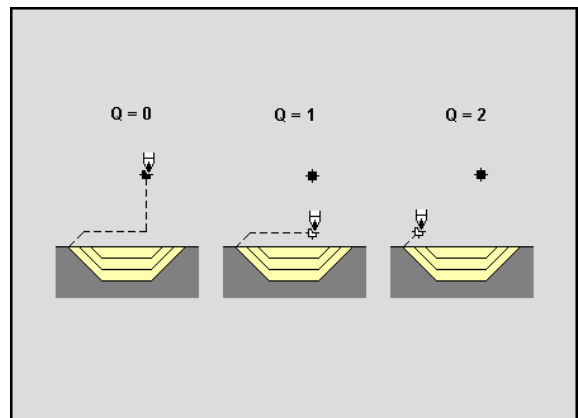
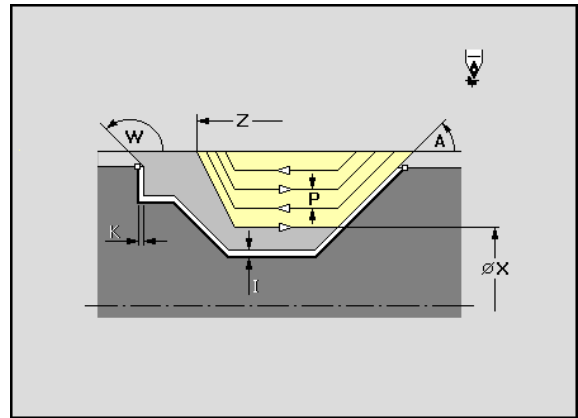
Bidirectional: Cutting with cycle

- ☐ Yes: G835
☐ No: G830

Q: Retraction at cycle end

- ☐ Q=0: Return to starting point
Longitudinal: first X, then Z direction
Transverse: First Z, then X direction
☐ Q=1: Positions in front of the finished contour
☐ Q=2: Lifts off to safety clearance and stops

Undercutting (see soft-key table)



6.12.5 Recessing

Overview of recessing operations

- Contour recessing (G860) – radial, axial or automatic
- Recessing (G866) – radial, axial or automatic
- Recess turning (G869) – radial, axial or automatic
- Parting (cutting off)
- Parting/prepare rear-side machining (workpiece transfer)

Contour recessing radial/axial (G860)

For form elements: Recess general, relief turns (recess type F) and freely defined recesses.

Parameters

X, Z: Cutting limit

Type of oversize is set by soft key

I, K: Different longitudinal/transverse oversize

I: Constant oversize – generates "oversize G58" before the cycle

Process: Setting by soft key

- Pre-cutting and finishing in one machining cycle
- Only pre-cutting
- Only finishing

Recessing radial/axial (G866)

For form elements: Recess type D (sealing ring), recess type S (guarding ring)

If an allowance has been specified, TURN PLUS first rough-machines and then finish-machines the recess. For finishing, the dwell period is accounted for only during the finishing operation. Otherwise, the dwell period is considered for each pass.

Parameters

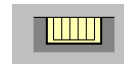
I: Oversize (longitudinal and transverse)

E: Period of dwell

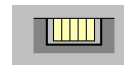
"Recessing sequence type" soft keys



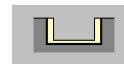
Select longitudinal/transverse oversize or constant oversize.



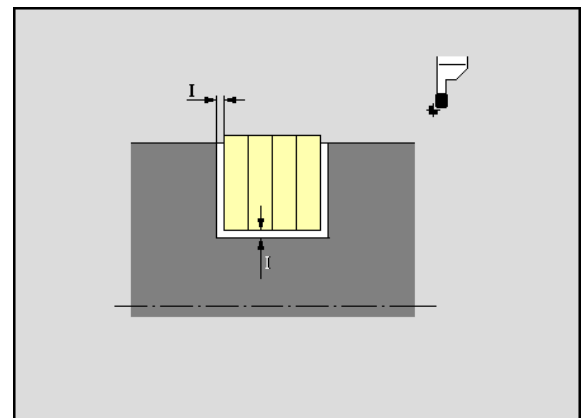
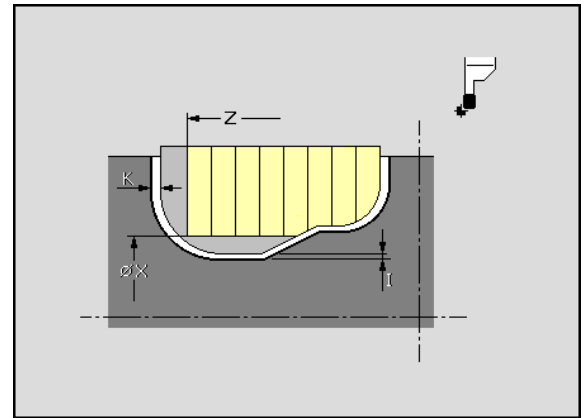
Precutting and finishing



Precutting



Finish-machining



Recess turning (G869)

The CNC PILOT machines the material using alternate recessing and roughing movements.

Parameters

- P: Maximum cutting depth
- R: Depth compensation - depending on factors such as workpiece material or feed rate, the tool tip "tilts" during a turning operation. You can correct this infeed error with "turning depth compensation factor" R. The compensation is usually determined empirically.
- B: Offset width – After the second infeed movement, during the transition from turning to recessing, the path to be machined is reduced by "offset width B." Each time the system switches from turning to recessing on this side, the path is reduced by "B" - in addition to the previous offset. After precutting, the remaining material is removed with a single cut.
- A, W: Approach angle, departure angle – reference: Z axis - default: opposite from the recessing direction
- X, Z: Cutting limit

Type of oversize is set by soft key

- I, K: Different longitudinal/transverse oversize
- I: Constant oversize – generates "oversize G58" before the cycle
- S: (Unidirectional/) bidirectional – setting by soft key
Precutting is performed:
- Yes (S=0): bidirectional
 - No (S=1): unidirectional in the direction defined during the selection of the machining area
- O: Recessing feed rate – default: Active feed rate
- E: Finishing feed – default: Active feed
- H: Type of retraction at cycle end
- H=0: Return to starting point (axial: first Z, then X direction; radial: first X, then Z direction)
 - H=1: Positions before the finished contour
 - H=2: Retracts to the safety clearance and stops

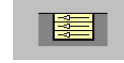
Process: Setting by soft key

- Pre-cutting and finishing in one machining cycle
- Only pre-cutting
- Only finishing

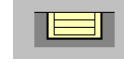
"Recess turning" soft keys



Select longitudinal/transverse oversize or constant oversize.



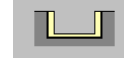
Unidirectional/Bidirectional



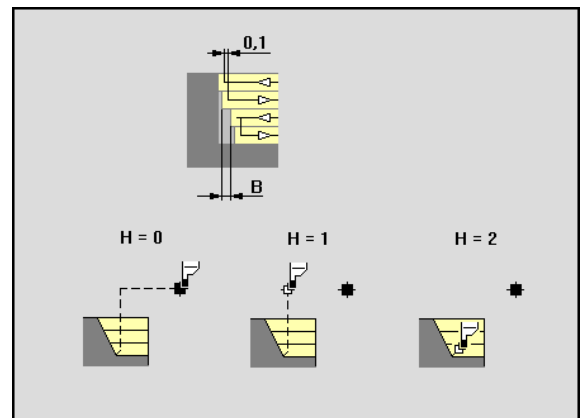
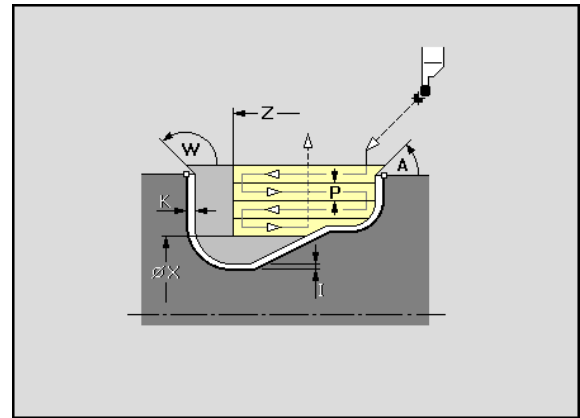
Precutting and finishing



Precutting



Finish-machining



Parting (Cutting-off)

The workpiece is parted with the **expert program** that is entered in machining parameter 21 – “UP 100098.” Expert programs are provided by the machine tool builder. That is why there may be deviations in the parameters described below. Use the expert program or the machine manual to inform yourself of the meaning of the parameters and the process of the expert program.

TURN PLUS determines the parameters as far as possible and enters them as default values. Check, edit or enter the values.

Parameters

Bar diameter (LA):

Starting point in Z (LB): TURN PLUS uses the position that you defined when selecting the machining range.

Chamfer/rounding (LC):

- < 0: Width of chamfer
- > 0: Radius of rounding

Feed-rate reduction from X (LD): Feed rate is reduced for the last path (the reduced feed rate is defined in the expert program).

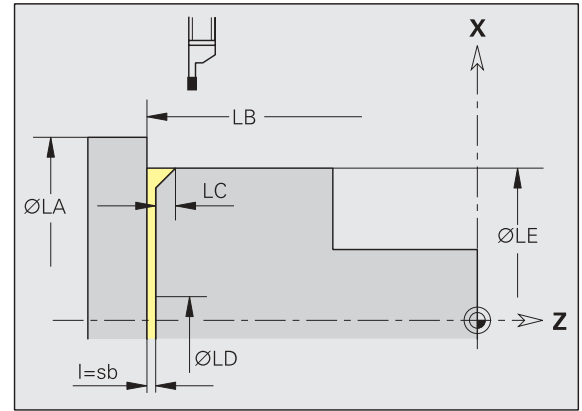
Diameter of finished part (LE): for determining the position of the chamfer/rounding

Inside diameter (LF): The expert program moves past this position in order to ensure a proper parting operation.

- = 0: for a solid bar
- > 0: for a tube

Safety clearance (LH): to starting position X

Cutting width (I): is usually not evaluated



Select the machining range:

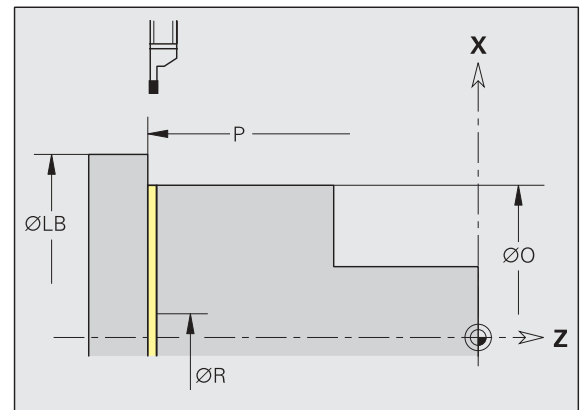
Vertical element, at which part is to be parted – and the chamfer/rounding.

Parting and tool transfer

TURN PLUS activates an **expert program** (from machining parameter 21) for parting and for workpiece transfer. Which expert program is used depends on the entry “1st setup of spindle .. – 2nd setup of spindle ..” in the program header:

- Same spindle (manual rechucking): Enter “UP-ABHAND”
- Different spindles (transfer of workpiece to the opposing spindle): Enter “UP-UMKOMPLA”

Expert programs are provided by the machine tool builder. That is why there may be deviations in the parameters described below. Use the expert program or the machine manual to inform yourself of the meaning of the parameters and the process of the expert program.



“Parting” parameters

Continued ►

Process of parting and workpiece transfer:

- ▶ Select the vertical element for the parting –TURN PLUS opens the dialog box of the expert program
- ▶ Check/edit the "parting" parameter
- ▶ The parting operation starts after OK is pressed
- ▶ Define the chucking data and position for the second setup
- ▶ Check/edit the "workpiece transfer" parameter
- ▶ The workpiece transfer starts after OK is pressed

TURN PLUS entered the calculated parameters as proposed values. Check, edit or enter the values.



The meaning of the transfer program depends on the name of the expert program.

Transfer parameters with the expert program "UMKOMPLA"

"Parting" (see sketch)

Spindle speed limitation (LA) for the parting process

Maximum diameter of workpiece blank (LB):

Proposed value: from the tool description

Reduced feed rate (K): for the parting operation

- 0: No feed rate reduction
- >0: (Reduced) feed rate

Starting point in X (O): For the parting operation – proposed value: from the workpiece description

Start point in Z (P): For the parting operation – proposed value: Vertical element from the "selection"

"Workpiece transfer" (see also "6.11 Prepare – Rechuck")

Speed or angular synchronism (LC):

- 0: Angular synchronism without angular offset
- >0: Angular synchronism with preset angular offset
- <0: Spindle speed synchronism

Transfer position in Z (LD):

- 0: Transfer position in machine dimension 1
- 1..6: Transfer position in machine dimension 1..6
- ≠ 0..6: Transfer position – Calculation of the proposed value: see sketch

Working position in Z (LE): Proposed value: Zero-point offset, such as from machine parameter 1164 for Z axis \$1

Finished part length (LF): From the workpiece description

Distance from stop surface (LH): Distance between reference point of chuck and stop surface of the clamping jaw, calculated from the second setup

Minimum traverse (I):

- Without moving to fixed stop: Safety clearance on the workpiece to be transferred – proposed value: From "safety clearance on blank" (machining parameter 2)
- With traverse to a fixed stop: See machine manual

Maximum traverse (J):

- No input: Without traverse to fixed stop
- With input: With traverse to fixed stop – meaning of the parameters I and J: See machine manual

1, if 2 workpieces (U): No meaning

Continued ►

Transfer parameters in expert program with other name**"Parting"** (see sketch)

Spindle speed limitation (LA) for the parting process

Feed rate reduction (LB): Feed value for the "last part" of the parting operation

Jaw rinsing (K): see machine manual

Starting position X (O): For the parting operation – proposed value: from the workpiece description

Position for reduced feed X (P): From this position, traverse is at reduced feed rate

End position X (R): End position during parting

Start position Z (S): For the parting operation – proposed value: Vertical element from the "selection"

Parting tool width (Y): Width of the parting tool's cutting edge

"Workpiece transfer" (see also "6.11 Prepare – Rechuck")

Angular synchronization (LC):

- ☐ 0: Angular synchronization
- ☐ 1: Speed synchronization

Angular offset (LD): with angular synchronization

Dead stop (LE):

- ☐ 0: With traverse to dead stop
- ☐ 1: Without traverse to dead stop

Machine dimension (LF): Transfer position in machine dimension n (n: 1..6)

Minimum feed path (LH): For "traversing to dead stop" (see machining manual)

Maximum feed path (LH): For "traversing to dead stop" (see machining manual)

Incr. feed path (J): For "traversing to dead stop" (see machining manual)

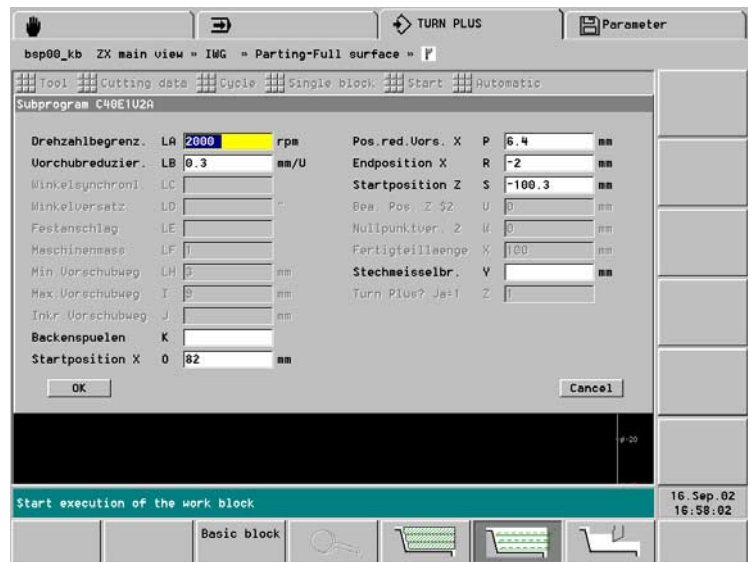
Working position Z \$2 (U): Working position of opposed spindle -proposed value: Zero point offset e.g. from machine parameter 1164 for Z axis \$1 (see sketch)

Zero point shift (W): Shift of the NC zero point (calculation: distance from reference point on chuck to dead stop on chuck jaw + finished part length)

Finished part length (LF): From the workpiece description

With TURN PLUS (Z):

- ☐ 1: Prepare work on the opposing spindle (switch-on conversions, zero point shift, etc.)



6.12.6 Drilling

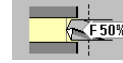
Overview of drilling operations

- Centric predrilling (G74)
- Centering (G72)
- Drilling (G71 or G74)
- Countersinking (G72)
- Counterboring (G72)
- Reaming (G71)
- Tapping (G73)
- Special drilling
 - Centering and countersinking (G72)
 - Drilling and countersinking (G72)
 - Drilling and threading (G73)
 - Drilling and reaming (G71 or G74)
- Drilling automatic – accounts for form elements, bore holes, single holes and hole patterns.

"Feed rate reduction" soft keys



Feed rate reduction for through drilling



Feed rate reduction for blind drilling

$bw < 180^\circ$

Feed rate reduction for blind drilling with boring bars and twist drills with 180° drilling angle

Centric predrilling (G74)

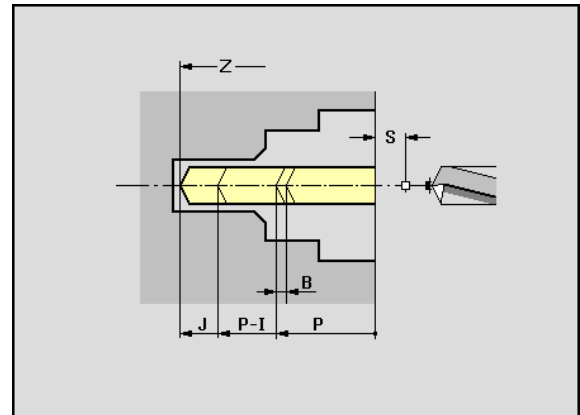
Predrilling at the turning center with stationary tools.

Selecting the machining range

Select all contour elements encompassing the bore hole. If required, the bore hole can be limited with "drilling limitation Z."

Parameters

- Z: Drilling limitation
- S: Safety clearance – generates "safety clearance G47" before the drilling cycle
- yP: 1st drilling depth
- J: Minimum drilling depth
- I: Reduction value
- B: Return distance – default: Retract to starting point of hole
- E: Period of dwell (for chip breaking at end of hole)



Position the drill with "Cycle – Approach" to the turning center.

Centric predrilling – Automatic

"Centric predrilling - Automatic" performs the complete predrilling operation, including tool changes that might be necessary because of different diameters.

Types of machining for holes

The IAG generates the following drilling and boring cycles:

- Centric predrilling: G74
- Centering: G72
- Drilling
- No deep-hole-drilling parameter set: G71
- deep-hole-drilling parameter set: G74
- Countersinking: G72
- Counterboring: G72
- Reaming: G71
- Tapping: G73
- Centering and sinking: G72
- Drilling and sinking: G72
- Drilling and thread: G73
- Drilling and reaming: G71 or G74

For

- Stationary tools: For drilling at the center of workpiece rotation
- Driven tools: For C-axis machining

Feed rate reduction factor

For blind drilling and/or through drilling you can define a feed rate reduction of 50%. The feed rate reduction for through drilling is switched on depending on the type of tool:

- Boring bar with indexable inserts and twist drills with 180° drilling angle: Feed-rate reduction switched on at drill tip – 2 * safety clearance
- Other tools: Tool tip – length of first cut – safety clearance (length of first cut = tool tip; safety clearance: see "Machining Parameter 9 Drilling or G47, G147")

Parameters

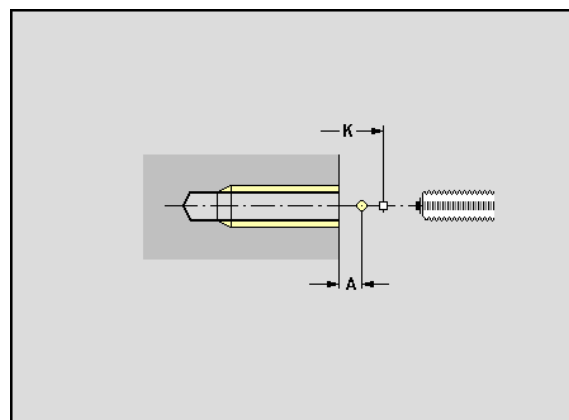
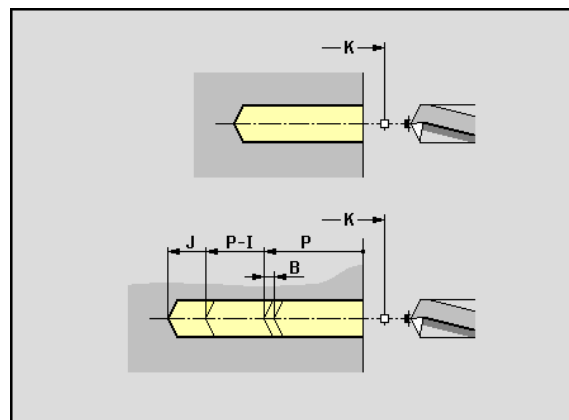
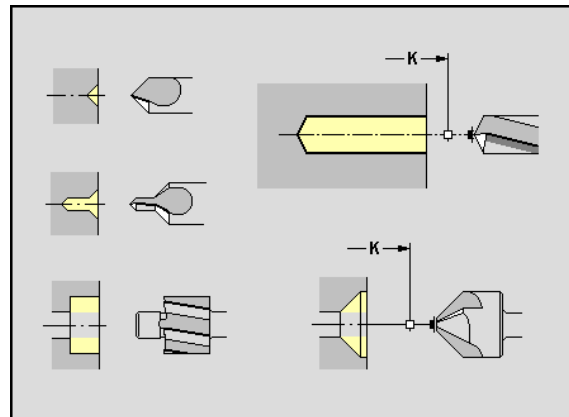
- K: Retraction plane – default: Return to the starting position or to the safety clearance
- D: Retract ("continue" soft key)
- At feed rate
 - At rapid traverse
- E: (Dwell time for) cutting free
- F50%: Feed rate reduction – see soft key table

Parameters (special deep-hole drilling)

- P: 1st drilling depth
- J: Minimum drilling depth
- I: Reduction of depth (reduction value)
- B: Retraction value (return distance) – default: Retract to starting point of hole

Parameters (special tapping)

- A: Slope length – default: Machining Parameter 7 "Thread starting length [GAL]"
- S: Retraction speed – default: Tapping speed



6.12.7 Finishing

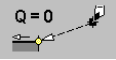
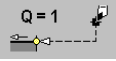
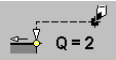
Overview of finishing operations

- Finishing – Contour machining (G890)
- Finishing - Clearance turning
- Finishing – Undercut
- Finishing – Residual-contour machining (G890 – Q=4)
- Hollowing finishing – neutral tool (G890 – Q=4)

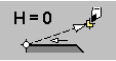
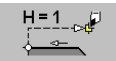

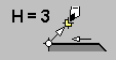
Notes on using TURN PLUS:

You define the "approach type, retraction type and form-element machining" by soft key – see the following tables.



"Finishing – Approach" soft keys

	<p>Approach: Automatic selection – the IWG checks:</p> <ul style="list-style-type: none"> ■ Diagonal approach ■ First X, then Z direction ■ Equidistant around the obstacle ■ Omission of the first contour elements, if the starting position is inaccessible
	Approach: First X, then Z direction
	Approach: First Z, then X direction














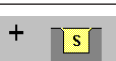


"Finishing - Retraction" soft keys

	Backs off at 45° against the machining direction and moves diagonally to the retraction position
	Backs off at 45° against the machining direction and moves first in the X direction, then in the Z direction to the retraction position
	Backs off at 45° against the machining direction and moves first in the Z direction, then in the X direction to the retraction position
	Backs off at feed rate to the safety clearance

"Form element machining" soft keys

	Switch over the soft-key row for the selection of the following form elements:
	Undercut type E

"Form element machining" soft keys

	Undercut type F
	Undercut type G
	Relief turn
	Switch over the soft-key row for the selection of the following form elements:
	Chamfer
	Rounding
	Fit
	Threads
	Switch over the soft-key row for the selection of the following form elements:
	Undercut type H
	Undercut type K
	Undercut type U
	Recess general
	Recess form S
	Recess form D
	Switch back the soft-key row

Finishing – Contour machining (G890)

The selected contour area is machined parallel to the contour in one pass. Chamfers, roundings, and undercuts are taken into account.

For **chamfers/roundings**, the following applies:

- The peak-to-valley height or feed rate are not programmed: The CNC PILOT automatically reduces the feed rate. At least "FMUR" revolutions (machining parameter 5) are performed.
- "Peak-to-valley height/feed rate" programmed: No feed rate reduction
- There is no feed-rate reduction for chamfers/rounding arcs that, due to their size, are machined with at least FMUR revolutions (machining parameter 5).

Parameters

X, Z: Cutting limit

Type of oversize is set by soft key

L, P: Different longitudinal/transverse oversize – generates "oversize G57" before the cycle

L: Constant oversize – generates "oversize G58" before the cycle

Plunging: Machine descending contours ?

- Yes
- No

E: Reduced plunging feed rate with descending contours

Approach:

- Yes: Set the type of approach Q by soft key
- No (Q=3): Tool is near the starting point

Q: Type of approach – define by soft key

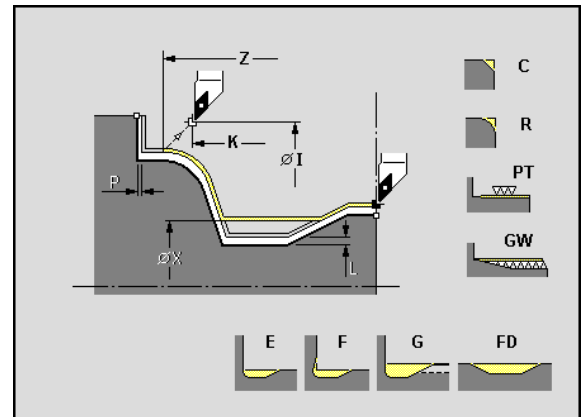
Retraction:

- Yes: Set the type of retraction by soft key
- No (H=4): Tool remains at the end coordinate

H: Type of retraction – define by soft key

I, K: Retraction position with H=0, 1 or 2

Form element machining with ...: Define by soft key the form elements, chamfers, etc. to be machined.



The CNC PILOT finds the proposed value of the "retraction position I,K" depending on whether you program "Cycle - Approach":

- programmed: Position from "Cycle - Approach"
- Not programmed: Position of the tool change point

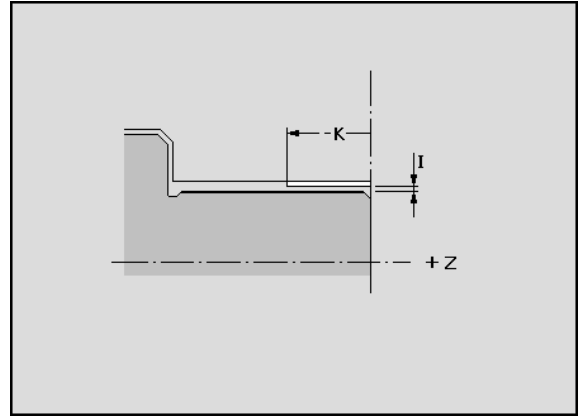
Finishing – Clearance turning

TURN PLUS executes a **measuring cut** on the selected contour element. Precondition: The "measuring" attribute was assigned to the contour element (see "6.9.6 Machining Attributes").

Parameters

- I: Oversize for measuring cut
- K: Length of measuring cut
- Q: Measuring loop counter (every nth workpiece is measured)

"Fit turning" is performed by the **expert program** (entry) "UP-MEAS01" (machining parameter 21). Parameters of the expert program: see your machine manual.



Finishing – Undercut

"Finishing – undercut" can be used for machining undercuts.

- Type U
- Type H
- Type K

When machining undercut type U, adjacent transverse elements for which an oversize is taken into account are machined to finished dimensions.

Notes on using TURN PLUS:

- ▶ Select the tool
- ▶ Selecting the machining range
- ▶ Confirm "Start"



You cannot influence an undercut operation; the "Cycle - Cycle parameter" menu item cannot be selected.

Finishing – Residual-contour machining (G890 – Q=4)

Using the "Roughing hollowing - residual roughing ." function, you can remove residual material from sloping contours.

Cutting limitation: The finishing operation begins with the residual material. Normally a cutting limit is not necessary.



Residual finishing (G890 – Q4) checks whether the tool can move into the contour valley without a collision. The collision check is based on tool parameter "width dn" (see "8.1.2 Tool Data").

Parameters

X, Z: Cutting limit

Setting the oversize type: by soft key

L, P: Different longitudinal/transverse oversize – generates "oversize G57" before the cycle

L: Constant oversize – generates "oversize G58" before the cycle

Plunging: Machine descending contours ?

■ Yes

■ No

E: Reduced plunging feed rate with descending contours

Retraction:

■ Yes: Set the type of retraction by soft key

■ No (H=4): Tool remains at the end coordinate

H: Type of retraction – define by soft key

I, K: Retraction position with H=0, 1 or 2

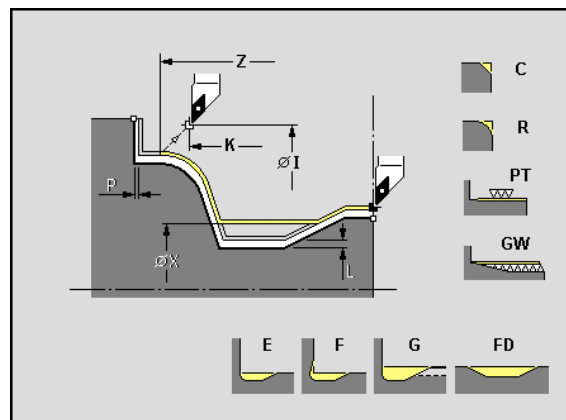
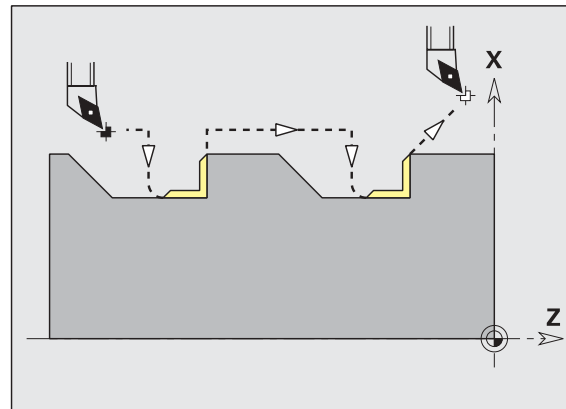
Form element machining with ...: Define by soft key the form elements, chamfers, etc. to be machined.



The CNC PILOT finds the proposed value of the "retraction position I,K" depending on whether you program "Cycle - Approach":

■ Programmed: Position from "Cycle – Approach"

■ Not programmed: Position of the tool change point



Finishing – Hollowing (neutral tool) (G890 – Q=4)

The IWG machines recess areas determined with the aid of the "inward copying angle" (recesses: inward copying angle \leq mtw).

During automatic generation, TURN PLUS selects a "neutral finishing tool"

Options (parameters)

X, Z: Cutting limit

Type of oversize is set by soft key

L, P: Different longitudinal/transverse oversize – generates "oversize G57" before the cycle

L: Constant oversize – generates "oversize G58" before the cycle

Plunging: Machine descending contours ?

- ☐ Yes
- ☐ No

E: Reduced plunging feed rate with descending contours

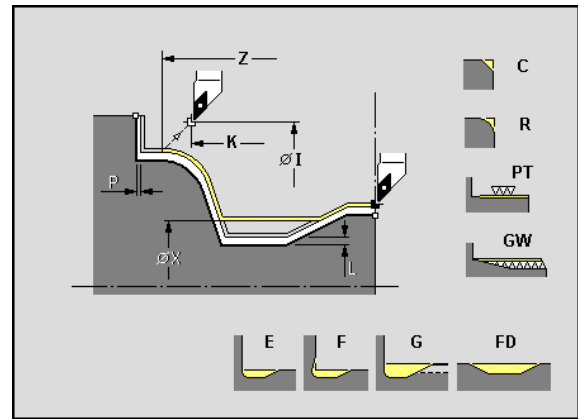
Retraction:

- ☐ Yes: Set the type of retraction by soft key
- ☐ No (H=4): Tool remains at the end coordinate

H: Type of retraction – define by soft key

I, K: Retraction position with H=0, 1 or 2

Form element machining with ...: Define by soft key the form elements, chamfers, etc. to be machined.

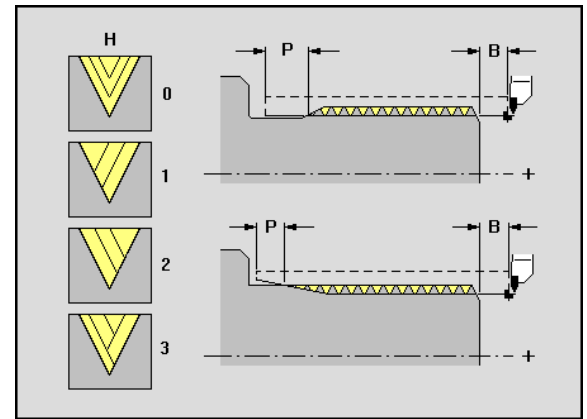


The position defined in "Cycle – Approach" is assumed as proposed value for the "Retraction position I, K"

6.12.8 Thread Machining(G31)

Parameters

- B, P: Starting length, overrun length - no input: The CNC PILOT automatically determines the length from adjacent undercuts or recesses. If no undercut/recess exists, the starting length and overrun length from machining parameter 7 will be used (see also "4.8 Thread Cycles").
- C: Starting angle - necessary when the starting point of the thread is defined with respect to contour elements that are not rotationally symmetric
- I: Maximum approach - maximum infeed distance
- V: Type of infeed
- Constant cross section (V=0): Constant cross section for all cuts
 - Constant approach (V=1)
 - Remaining cut division (V=2): If the division thread depth/infeed leaves a remainder, the first feed is reduced. The "last cut" is divided into 1/2, 1/4, 1/8 and 1/8 of a cut.
 - EPL method (V=3): The infeed is calculated from the pitch and the speed
- H: Type of tool offset of the individual infeeds to smooth the thread flanks
- H=0: No offset
 - H=1: Offset from left
 - H=2: Offset from right
 - H=3: Offsets alternates right/left
- Q: Number of air cuts – after the last cut (for reducing the cutting pressure in the thread base)



Danger of collision!

An excessive overrun length P might cause a collision. The overrun length can be checked during the simulation.

6.12.9 Milling

Overview of milling operations

- Contour milling – roughing, finishing (G840)
- Area milling – roughing (G845), finishing (G846)
- Deburring (G840)
- Engraving (G840)
- Automatic milling – roughing, finishing

Contour milling – roughing/finishing, deburring (G840)

Contour milling and deburring are for figure or “free contours” (open or closed) of the reference planes:

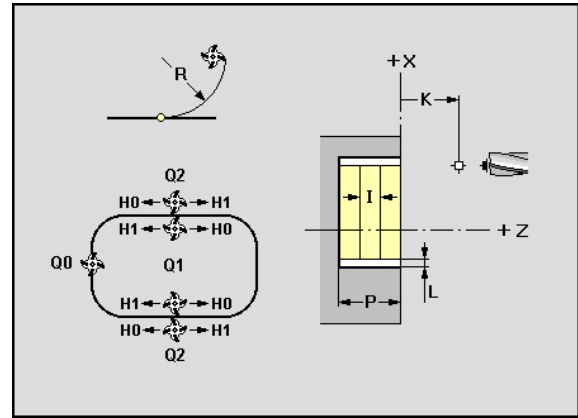
- FRONT
- REAR SIDE
- SURFACE

The **oversize L** “shifts” the milling contour in the direction defined under “milling location Q”:

- Q=0: Oversize is ignored
- Q=1 (closed contour): Reduces the size of the contour
- Q=2 (closed contour): Enlarges the contour
- Q=3 (open contour): Shift left/right – depending on the machining direction

Parameters

- K: Retraction plane – default: return to starting position
- Front/rear face: Retraction position in Z direction
 - Lateral surface: Retraction position in X direction (diameter)
- Q: Milling location
- Q=0 Contour: Cutter center on the contour
 - Q=1 inside (milling) – closed contour
 - Q=2 Outside (milling) – closed contour
 - Q=3 left/right of the contour (reference: machining direction) – open contour
- H: Cutting direction
- H=0: Up-cut milling
 - H=1: Climb milling
- R: Approach radius
- R=0: Directly approach contour element
 - R>0: Approaching/departing radius connecting tangentially with the contour element
 - R<0 with inside corners: Approach/departure radius connecting tangentially with the contour element
 - R<0 with outside contours: Contour element is approached/departed tangentially on a line



Continued ►

- P: ■ Contour milling: Milling depth – overwrites the “depth” of the contour definition
 ■ Deburring: Plunging depth of the tool – default: Chamfer width (from “deburring” machining attribute) + 1 mm
- I: Maximum infeed – default: Milling in one infeed
- L: Oversize – “Shift” milling contour (“oversize G58” before the milling cycle)



- Effects of “milling location, cutting direction and direction of tool rotation”: See “4.11 Milling Cycles.”
- **Deburring:** The **chamfer width** is defined as machining attribute.

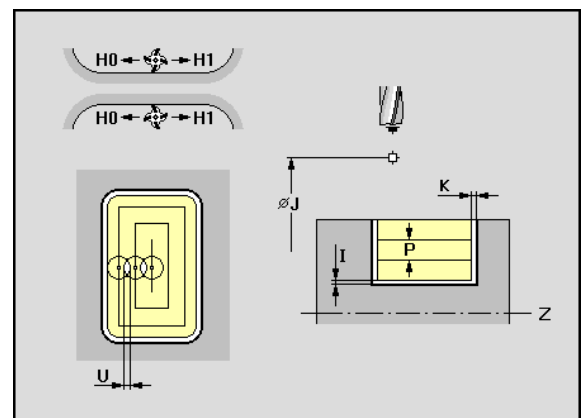
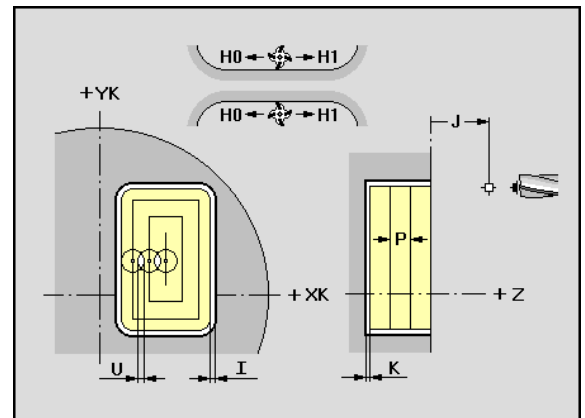
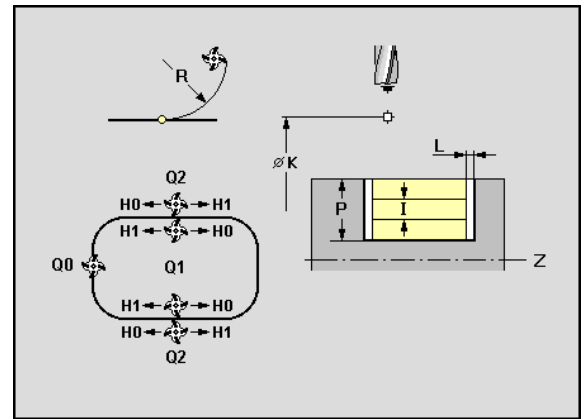
Area milling - Roughing/Finishing (G845/G846)

Roughs/finished figures or closed “free contours” of the reference planes:

- FRONT
 ■ REAR SIDE
 ■ SURFACE

Parameters

- J: Retraction plane – default: return to starting position
 ■ Front/rear face: Retraction position in Z direction
 ■ Lateral surface: Retraction position in X direction (diameter)
- Q: Machining direction (Q)
 ■ Toward the outside (Q=0): From the inside toward the outside
 ■ Toward the inside (Q=1): From the outside toward the inside
- H: Cutting direction
 ■ H=0: Up-cut milling
 ■ H=1: Climb milling
- U: Overlap factor – range: $0 \leq U \leq 0.9$;
 0: No overlapping
- V: Overshoot factor – has no effect for operations with the C axis
- P: Maximum infeed in the milling plane
- I, K: Oversize in X, Z direction - is omitted for finishing



Engraving (G840)

Engraves open or closed contours of the reference plane:

- FRONT
- REAR SIDE
- SURFACE

Options (parameters)

- K: Retraction plane – default: return to starting position
- Front/rear face: Retraction position in Z direction
 - Lateral surface: Retraction position in X direction (diameter)
- P: Milling depth – plunging depth of the tool

6.12.10 Special Machining Tasks (SM)

In **special machining (SM)** you add paths of traverse, subprogram calls or G/M functions (example: use of tool handling systems).

“Special machining” defines a work block that is integrated in the working plan.

Special machining tasks

- Tool movements at feed rate or rapid traverse, including tool calls and definition of technological data

Selection:

- ▶ IWG menu item “Special mach(ining)”
- ▶ Menu item “free input”
- ▶ Menu item “Tool” – Select and position the tool
- ▶ Select the “Single block” menu item
- ▶ To define the tool path and other technological data (G/M functions), use the other menu items
- Subprogram call, G and M functions
 - ▶ Select the “SM” drop-down menu.
 - ▶ Choose “Free input”
 - ▶ Select the “Single block” menu item
 - ▶ Select the “Technology” drop-down menu
 - ▶ Select the menu item “Subprogram” or “G and M functions”
 - ▶ Select the desired subprogram/function. Confirm with “OK”

6.13 Automatic Working Plan Generation (AWG)

The **AWG** generates a **working plan** consisting of individual **working blocks**. TURN PLUS automatically finds the elements of a working block. The "control graphics" feature enables you to directly check a working block (see "6.14 Control Graphics").

The machining sequence can be influenced through the **Machining Sequence Editor** (see "6.13.2 Machining Sequence").

The AWG enables you to continue machining a partially machined workpiece.

6.13.1 Generating a Machining Plan

Selection: AWG – Automatic

TURN PLUS generates the working blocks according to the sequence defined in "Machining order" and displays them in the control graphics. After generation you can

- ☐ Accept or
- ☐ Reject the working plan.

To **interrupt** working plan generation, press the ESC key. All working blocks that have **completely** been generated up to the moment of interruption are retained.

Selection: AWG – Block-by-block

TURN PLUS generates the working blocks according to the sequence defined in "Machining order" and displays them in the control graphics. After generation you can

- ☐ accept,
- ☐ reject or
- ☐ repeat the working block.

After blockwise **working plan** generation you can

- ☐ Accept or
- ☐ Reject the working plan.

TURN PLUS uses default values for machining details that cannot be defined through contour analysis, attributes, etc. TURN PLUS displays a warning for your information. Example: If you haven't defined how to clamp a workpiece, TURN PLUS assumes a default type of clamping/clamping length and adjusts the cutting limitation accordingly.



6.13.2 Machining Sequence

TURN PLUS analyzes the contour in the sequence in which the operations are listed. In this process the areas to be machined and the tool parameters are ascertained. The contour is analyzed with the aid of the machining parameters.

TURN PLUS differentiates in machining operations between:

- Main machining operations
- Submachining operations
- Location (machining location)

With the "submachining" and the "machining location" you can refine the machining specification. If you do not define the submachining operation/ machining location, the AWG generates working blocks for **all** submachining operations/ machining locations.

The following table lists the recommended combinations of main machining operations with submachining operations and machining locations and explains the working method of the AWG.

The following factors additionally influence the working plan:

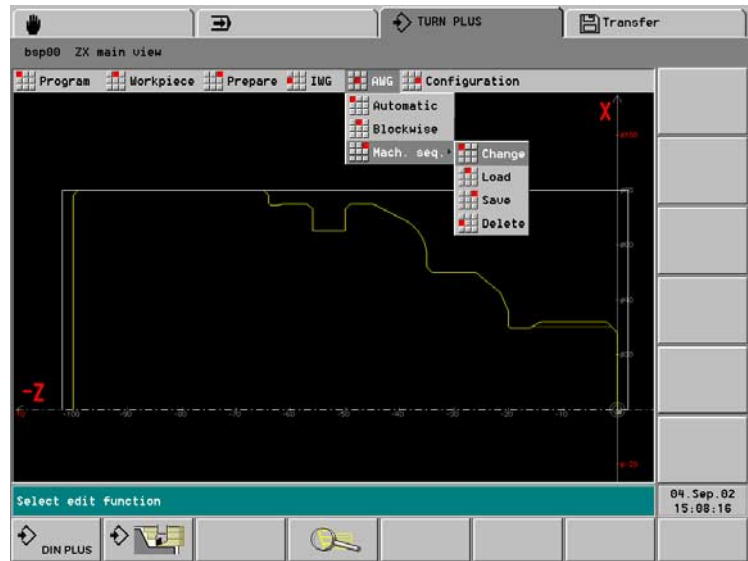
- Contour geometry
- Contour attributes
- Tool availability
- Machining parameters

The AWG does **not** generate the working blocks if any required preparatory step is missing, or if the appropriate tool is not available, etc. TURN PLUS skips machining operations/ machining sequences that do not make sense in the machining process.

Rear-face machining (full-surface machining)

Rear-face machining is initiated by the main machining and submachining operation "Cutting off – Full-surface machining" or by "Rechucking - Full-surface machining."

- You can define further machining operations for machining the rear face after "Parting ... / Rechucking ..."
- When no further main machining operations are defined after "Cutting off ... / Rechucking ...," TURN PLUS uses the machining sequence defined for the front face for machining the rear-face.



- TURN PLUS always uses the **current machining sequence**. The current machining sequence can be edited or overwritten by loading another machining sequence.
- If you load a "complete" program and generate a new working plan, the **current machining sequence** is used as basis.



Danger of collision!





When executing drilling or milling operations, TURN PLUS does not check whether the turning operation has already been completed. Ensure that turning operations precede drilling or milling operations in the machining sequence.


Continued ►

List of Machining Sequences


Main machining	Submachining	Location	Execution
Centric predrilling			Contour analysis: Determining the drilling steps Machining parameter: Centric predrilling (3)
	–	–	Predrilling, 1st step Predrilling, 2nd step Finish-drilling
	Predrilling	–	Predrilling, 1st step Predrilling, 2nd step
	Finish-drilling	–	Finish-drilling
Roughing (without hollowing)			Contour analysis: Subdivision of the contour into areas for external longitudinal/external transverse and internal longitudinal/internal transverse machining, using the transverse/longitudinal ratio (PLVA, PLVI). Sequence: External machining precedes internal machining. Machining parameter: Roughing (4)
	–	–	Transverse machining, longitudinal machining - inside and outside
	Longitudinal	–	Longitudinal machining – outside and inside
	Longitudinal	Outside	Longitudinal machining – outside
	Longitudinal	Inside	Longitudinal machining – inside
	Transverse	–	Transverse machining
	Contour-parallel	–	Contour-parallel machining – outside and inside
	Contour-parallel	Outside	Contour-parallel machining – outside
	Contour-parallel	Inside	Contour-parallel machining – inside
(Roughing) Hollowing			Contour analysis: Using the “inward copying angle (EKW),” you can determine recess areas (undefined recesses). The workpiece is machined with one or two tools. Sequence: External machining precedes internal machining. Machining parameters: Global finished-part parameters (1)
	–	–	Longitudinal/transverse machining – outside and inside
	Longitudinal	Outside	Longitudinal machining – outside
	Longitudinal	Inside	Longitudinal machining – inside


Continued ►

Main machining	Submachining	Location	Execution
(Roughing) Hollowing – continued			
	Transverse	Outside	Transverse machining - outside, front and back
	Transverse	Inside	Transverse machining - inside
	Transverse	Outside/front	Transverse machining - outside/front
	Transverse	Outside/back	Transverse machining - outside/back
	Neutral tool	–	Longitudinal/transverse machining – outside and inside
	Neutral tool	Outside	Longitudinal machining – outside
	Neutral tool	Inside	Longitudinal machining – inside
	Neutral tool	Outside/front	Transverse machining - outside, front and back
	Neutral tool	Inside/front	Transverse machining - inside
			 If hollowing precedes recess-turning/contour-recessing in the machining sequence, recess areas are hollowed – exception: no suitable tools are available.
Contour machining (finishing)			
			Contour analysis: Subdivision of the contour into areas for external and internal machining. Sequence: External machining precedes internal machining. Machining parameter: Finishing (5)
	Contour-parallel	–	External/internal machining
	Contour-parallel	Outside	External machining
	Contour-parallel	Inside	Internal machining
	Neutral tool	–	External/internal machining
	Neutral tool	Outside	External machining
	Neutral tool	Inside	Internal machining
	Neutral tool	Outside/front	Machining the front/back - outside
	Neutral tool	Inside/front	Machining the front - inside
			 Undefined recesses are only finish-machined if they have been rough-machined before.  Contour-parallel submachining (standard tools): Finishing operation according to hollowing operation.  Submachining with neutral tool: Finish-machining using one tool.


Main machining	Submachining	Location	Execution
Recess turning			Contour analysis: ■ Without previous rough-machining : The complete contour, including recess areas (undefined recesses), is machined. ■ With previous rough-machining : Recess areas (undefined recesses) are determined from the "inward copying angle EKW" and machined. Sequence : External machining precedes internal machining. Machining parameters : Global finished-part parameters (1)
	–	–	Radial/axial machining – outside and inside
	Contour-parallel	Outside	Radial machining – outside
	Contour-parallel	Inside	Radial machining – inside
	Contour-parallel	Outside/front	Axial machining – outside
	Contour-parallel	Inside/front	Axial machining – inside
<div> ■ If recess turning precedes hollowing in the machining sequence, recess areas are machined by recess turning. – exception: no suitable tools are available. ■ Recess turning – contour turning are used alternatively.</div>			
Contour recessing (contour cutting)			Contour analysis : Recess areas (recesses) are determined from the "inward copying angle EKW" and machined. Sequence : External machining precedes internal machining. Machining parameters : Global finished-part parameters (1)
	–	–	Radial/axial machining – outside and inside Shaft machining: Axial machining on the outside is executed on the front and back.
	Contour-parallel	Outside	Radial machining – outside Shaft machining: front and back
	Contour-parallel	Inside	Radial machining – inside
	Contour-parallel	Outside/front	Axial machining – outside

Continued ►

Main machining	Submachining	Location	Execution
Contour recessing – continued			
	Contour-parallel	Inside/front	Axial machining – inside
			<div><ul style="list-style-type: none">■ If contour recessing precedes hollowing in the machining sequence, recess areas are machined by contour recessing. – exception: no suitable tools are available.■ Recess turning – contour turning are used alternatively.</div>
Recessing			
			Contour analysis: Determining the recess type: <ul style="list-style-type: none">■ Form element S (guarding ring – recess type S)■ Form element D (sealing ring – recess type D)■ Form element A (recess general)■ Form element FD (relief turn F) - for FD, a recess operation is used only in conjunction with "inward copying angle EKW + mtw (contour recessing angle). Sequence: Outside machining before inside machining machining parameters for form element FD: Global finished-part parameters (1)
	–	–	All recess types; radial/axial machining; outside and inside
	Type S, D, A, FD (*)	Outside	Radial machining – outside
	Type S, D, A, FD (*)	Inside	Radial machining – inside
	Type A, FD (*)	Outside/front	Axial machining – outside
	Type A, FD (*)	Inside/front	Axial machining – inside
	*: Define the recess type		
Undercuts			
			Contour analysis/machining: Determining the undercut types: <ul style="list-style-type: none">■ Form element H is machined using individual traverse paths; copying tool (type 22x)■ Form element K is machined using individual traverse paths; copying tool (type 22x)■ Form element U is machined using individual traverse paths; Recessing tool (type 15x)■ Form element G is machined with cycle G860 Sequence: External machining precedes internal machining; radial machining precedes axial machining.
	–	–	All recess types; outside and inside
	Type H, K, U, G (*)	Outside	Machining, outside

Main machining	Submachining	Location	Execution
Undercuts – continued			
	Type H, K, U, G (*)	Inside	Machining, inside
	*: Define the undercut type		
			<div> TURN PLUS uses roughing and finishing operations to machine undercut type G. Undercut type G is only machined with an undercut cycle if no suitable roughing/finishing tool is available.</div>
Thread cutting			
	–	–	Contour analysis: Determining the thread types. Sequence: External machining precedes internal machining. Then the elements are machined according to the sequence of geometrical definition.
	Cylindric (longitudinal)		Machining cylindric (longitudinal), tapered and transverse threads on the outside and inside of a contour.
	Tapered, transverse (*)	Outside	Machining an external thread
	Cylindric (longitudinal)		
	Tapered, transverse (*)	Inside	Machining a threaded hole
	*: Define the type of thread		
Drilling			
			Contour analysis: Determine the type of bore hole (form elements). Sequence – drilling operation/drilling combinations: ■ Centering / centering & countersinking ■ Drilling ■ Countersinking / drilling & countersinking ■ Reaming / drilling & reaming ■ Tapping / drilling & threading Sequence – machining location: ■ Center ■ Front face (also for Y front face) ■ Lateral surface (also for Y lateral surface) – following that, sequence of geometrical definition.
	–	–	Machining all bore holes on all machining locations
	Centering, drilling, Countersinking, reaming, tapping (*)	–	Execution of the selected drilling operation on all machining locations

Continued ►

Main machining	Submachining	Location	Execution
Drilling – continued			
	Centering, drilling, Countersinking, reaming, tapping (*)	Location	Machining a bore hole on the selected machining location
	*: Define the drilling cycle		
			 Drilling combinations: <ul style="list-style-type: none"> ■ Define the drilling combinations as machining attributes (see "6.9.6 Machining Attributes"). ■ Select the appropriate drilling operation as submachining operation (see above).
Milling			
			Contour analysis: Determining the milling contour. Sequence – milling operation: <ul style="list-style-type: none"> ■ Linear and circular slots ■ Open contours ■ Closed contours (pockets), single surfaces and polygonal surfaces Sequence – machining location: <ul style="list-style-type: none"> ■ Front face (also machines front face in Y) ■ Lateral surface (also machines lateral surface in Y) – then sequence of geometrical definition
	–	–	Execution of all milling operations on all machining locations
	Surface, contour, slot, pocket (*)	–	Execution of the selected milling operation on all machining locations
	Surface, contour, slot, pocket (*)	Location	Execution of the selected milling operation on the selected machining location
	*: Define the type of contour		

Main machining	Submachining	Location	Execution
Deburring			Contour analysis: Determining milling contours with the attribute "Deburring." Sequence – machining location: <ul style="list-style-type: none">■ Front face (also machines front face in Y)■ Lateral surface (also machines lateral surface in Y) – then sequence of geometrical definition
	–	–	Machining all milling contours with the attribute "Deburring" on all machining locations
	Contour, slot, pocket (*)	Location	Machining all milling contours with the attribute "Deburring" on the selected machining location
*: Define the type of contour			
Engraving			Contour analysis: Determining milling contours with the attribute "Engraving." Sequence – machining location: <ul style="list-style-type: none">■ Front face (also machines the front face in Y)■ Lateral surface (also machines the lateral surface in Y) – then sequence of geometrical definition
	–	–	Machining all milling contours with the attribute "Engraving" on all machining locations
	Contour, slot (*)	Location	Machining all milling contours with the attribute "Engraving" on the selected machining location
*: Define the type of contour			
Finish-milling			Contour analysis: Determining the milling contour. Sequence – milling operation: <ul style="list-style-type: none">■ Linear and circular slots■ Open contours■ Closed contours (pockets), single surfaces and polygonal surfaces Sequence – machining location: <ul style="list-style-type: none">■ Front face (also machines front face in Y)■ Lateral surface (also machines lateral surface in Y) – then sequence of geometrical definition
	–	–	Execution of all milling operations on all machining locations
	Surface, contour, slot, pocket (*)	–	Execution of the selected milling operation on all machining locations

Continued ►

Main machining	Submachining	Location	Execution
Finish-milling – continued			
	Surface, contour, slot, pocket (*)	Location	Execution of the selected milling operation on the selected machining location
	*: Define the milling operation		
Parting (Cutting-off)	–	–	The workpiece is cut off.
	Full-Surface Machining	–	The workpiece is cut off and transferred to the counterspindle.
Rechucking	Full-Surface Machining	–	<ul style="list-style-type: none"> ■ Lathe with counterspindle: The workpiece is transferred to the counterspindle. ■ Lathe fitted with one spindle: The workpiece is rechucked manually.
Machining of fits	For contour machining (finishing), the AWG takes into account contour elements with the "measuring" contour element.		
Special machining tasks	No significance for the AWG		

Editing and managing machining sequences

Editing a machining sequence

Select "AWG - Machining order - Change";TURN PLUS opens the machining sequence editor.

Select the position

Entering a new machining sequence

- Position the cursor (the new code is inserted in front of the cursor position)

Insert

Activates the "Enter the machining sequence" dialog. Select

- Main machining
- Submachining
- Location

and confirm with "Enter."

"OK" confirms the new machining sequence

Editing the machining sequence

- Position the cursor

Change

Activates the "Enter the machining sequence" dialog. Select

- Main machining
- Submachining
- Location

and correct it with "Enter."

"OK" confirms the edited machining sequence

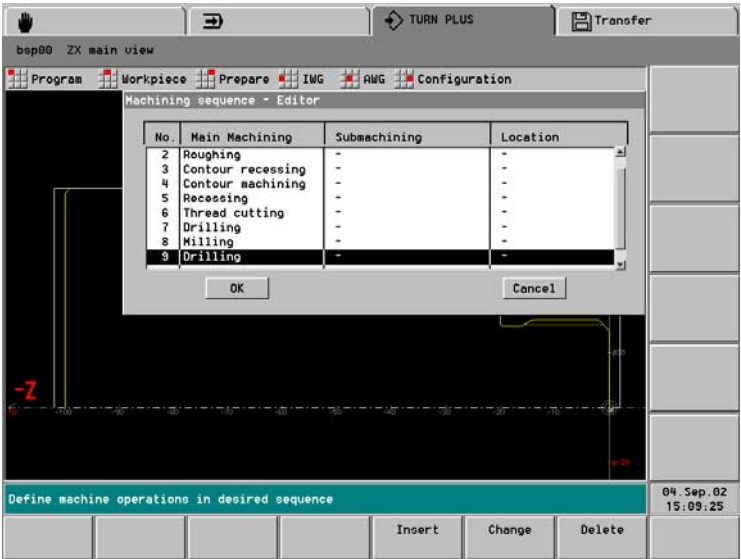
Deleting a machining operation

- Position the cursor

Delete

TURN PLUS removes the NC code

"OK" saves the changes machining sequence



Managing the machining sequence files

The following subpoints of "AWG – Machining sequence" help you manage the files:

- Load
- Saving (storing on hard disk)
- Delete

6.14 Control Graphics

During **contour definition**, TURN PLUS displays all contour elements that can be displayed.

The **IWG** and **AWG** permanently display the finished part contour and graphically depict the cutting operations. The workpiece blank **takes on a contour** during machining.

You can adjust the depiction of the **tool paths** and the **simulation mode** by soft key.

Maximum window size

If there is more than one window on the screen, you can use the " key to switch windows settings from full-size to multiple windows.

Zoom



After activation, a red frame appears with which you can select the detail you wish to isolate. The submenu "Zoom standard settings" also appears.

Zoom settings by keyboard

- Enlarge: "Page forward"
- Reduce: "Page back"
- Shift: Cursor keys

Zoom settings by touch pad

- Position the cursor to one corner of the section
- While holding the left mouse key, pull the cursor to the opposite corner of the section

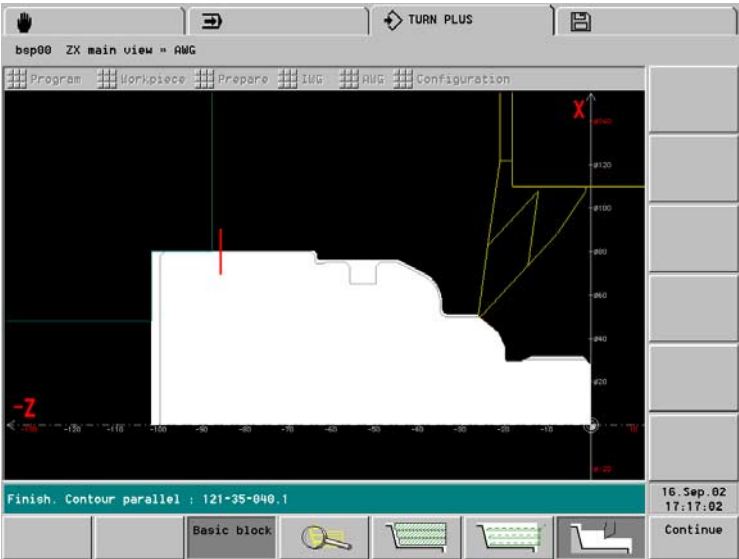
Standard settings: See soft-key table

After having enlarged a detail to a great extent, select "Workpiece maximum" or "Work space," and then isolate a new detail.

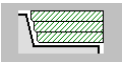
To exit the zoom function, press the ESC key.

"Control graphics" soft keys

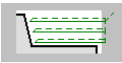
Basic block	■ On: stops after every traverse movement
	■ Off: simulates the complete machining sequence
Continue	Run the next path of traverse ("basic block on" simulation mode)
	Activate the zoom function



"Control graphics" soft keys



(Cutting) path: depicts the surface traversed by the "cutting area" of the tool with hatch marks.



Line: Depicts paths of traverse with solid line (reference: theoretical tool tip)

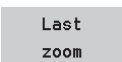


Material-removal graphic: "cuts" (erases) the area traversed by the "cutting area" of the tool.

Zoom soft keys



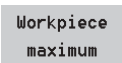
Standard size Shows the last setting "workpiece maximum" or "working space."



Last zoom Cancels the last magnification/setting. You can select "Previous zoom" more than once.



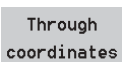
Switches the zoom function to the next window.



Workpiece maximum Shows the workpiece in the largest possible view.



Work space Shows the working space including the tool change position.



Through coordinates Define the coordinate system and the position of the workpiece zero (see "6.15 Configuration")

6.15 Configuration

With the functions of the "configuration" you change and manage various display and input variants.

Settings:

Zoom behavior:

- Dynamic: Adjusts the the contour depiction to the window size
- Static: Adjusts the contour depiction to the window size when the contour is loaded and keeps this setting

Plane ID (designation of the coordinate axes)

- Display
- Do not display

Point grid (in background)

- Display
- Do not display

X value input (for basic elements and form elements of the turning contour)

- Diameter: Input is interpreted as diameter values
- Radius: Input is interpreted as radius values

With help graphic (to illustrate the input parameters)

- Yes: Display help graphics
- No: Do not display help graphics

Window configuration ("Views" menu item):

Views that TURN PLUS is to depict besides the main view (XZ plane) (front face, unrolled lateral surface, etc.).

Mirror main view ?

- Yes: Complete contour is depicted
- No: contour above the turning center is shown

Coordinates:

Setting of the coordinate system and the position of the workpiece datum for the

- Main view
- Front face
- Rear side
- Lateral surface

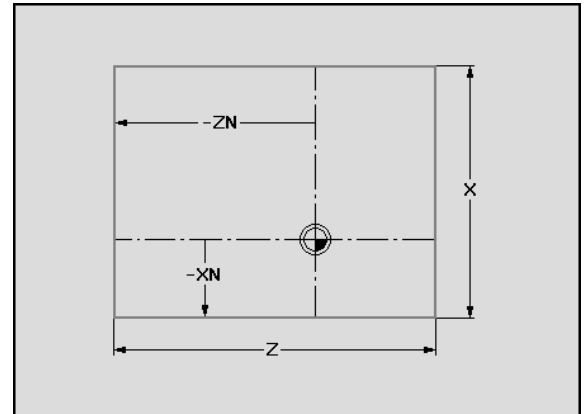
Parameters (example of main view)

Delta X, Z: defines the dimensions of the control-graphic window

min XN, ZN: Defines the position of the tool zero point



X value input: With standard forms for workpiece blank description, X values always function as diameter values. X/XE coordinates on contours for C/Y axis machining always function as radius values.



TURN PLUS

- Adjusts the dimensions to the height and width of the screen.
- Increases the dimensions of the window to show the complete workpiece.

Control graphic

Depending on the menu option selected (IWG or AWG), the following settings apply to the IWG or AWG:

Basic block:

- On: stops after every traverse movement
- Off: simulates the complete machining sequence

Graphic type:

- Tool path: depicts paths of traverse with solid line (reference: theoretical tool tip)
- Cutting path: depicts the surface covered by the "cutting area" of the tool with hatch marks. The cutting path graphic accounts for the exact geometry of the tool tip (cutting radius, cutting width, tool-tip position, etc.). The graphic simulation is based on the tool data.
- Erasing graphics: The blank part is displayed as a "filled area" from which material is removed during the machining process.

6.16 Machining Information

6.16.1 Tool Selection, Turret Assignment

The tool selection is determined by:

- The machining direction
- The contour to be machined
- The machining sequence

If the ideal tool is not available,

- TURN PLUS first searches for a replacement tool,
- then for an "emergency" tool.

If necessary, TURN PLUS adapts the machining cycle to the requirements of the replacement or emergency tool. If more than one tool is suitable for a machining operation, TURN PLUS uses the optimal tool.

TURN PLUS does not support multiple tools (except tools for drilling combinations).

Contour recessing, recess turning

Cutting radius must be smaller than the smallest inside radius of the recess contour, but ≥ 0.2 mm.

TURN PLUS determines the width of the recessing tool as follows:
The recess contour contains

- Paraxial base element with radii on both sides:
 $SB \leq b + 2 \cdot r$ (for different radii: smallest radius)
- Paraxial base element without radii or with a radius on one side only: $SB \leq b$
- No paraxial base element: The recessing width is determined using the "recessing width divisor (SBD)" (machining parameter 6)

SB: Recessing width

b: Width of base element

r: Radius

Drilling

Depending on the geometry of the bore hole, TURN PLUS determines the appropriate tool. For centric bore holes, TURN PLUS uses stationary tools.

Automatic turret assignment

The tool location is selected according to the "Mount type" and "Preferred tool" parameters (machine parameter 511, ...). These parameters are used to determine whether a driven tool is supported and whether to locate primarily external/internal tools or drilling/milling tools.

Location type

The "mount type" (machine parameter 511, ...) differentiates between different tool holders (see "8.1.2 Notes on Tool Data").

TURN PLUS does not support magazine pocket systems.



Machining parameter 2 (global technology parameters) defines whether TURN PLUS is to account for the tool database in addition to the current turret assignment.

6.16.2 Cutting Parameters

To determine the cutting parameters, TURN PLUS uses the

- Workpiece material (program head)
- Cutting material (tool parameters)
- Machining type (for the IWG: primary machining operation selected; for the AWG: primary machining operation from the machining sequence)

The values determined are multiplied by the correction factors which depend on the respective tool (see "8.3 Database for Technology Parameters (Cutting Values)" and "8.1.2 Tool Data").

Note for roughing and finishing operations:

- Main feed rate when the main cutting edge is used.
- Secondary feed rate when the secondary cutting edge is used.

For milling operations, the following applies:

- Main feed rate for operations in the milling plane
- Secondary feed rate for infeed movements

For threading, drilling and milling operations, the cutting speed is converted into rotational speed.

6.16.3 Coolant

Depending on the workpiece material, cutting material and machining operation, define in the technology database whether coolant is used.

AWG

If you have specified that coolant is to be used, the AWG activates the coolant circulation for the respective machining block. If high-pressure coolant circulation is used, the AWG generates a corresponding M function.

When a fixed turret assignment (see machining parameter 2) is used, each tool can be assigned high-pressure/normal-pressure coolant circulations (selection: "Prepare - Tool list – Set up list"). The AWG activates the respective coolant circulations as soon as the tool is used.

IWG

The IWG controls coolant circulation in the same way as the AWG. Alternately, you can define coolant circulation and pressure stage for the current machining block in the cutting data.

6.16.4 Hollowing

If hollowing precedes recess-turning or contour-recessing in the machining sequence, recess areas (undefined recesses) are machined with roughing tools. Otherwise, the AWG machines the respective contour areas using recessing tools. TURN PLUS uses the "Inward copying angle EKW" (machining parameter 1) to distinguish recesses from relief turns.

If more than one tool is required for the hollowing operation, TURN PLUS pre-machines the area with the first tool and removes the residual material with a tool machining in the opposite direction.

Contour machining (finishing): The AWG finish-machines hollowed recesses using the same strategy as for the roughing operation.

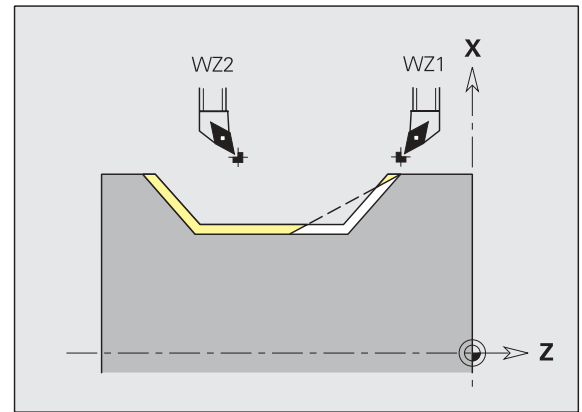
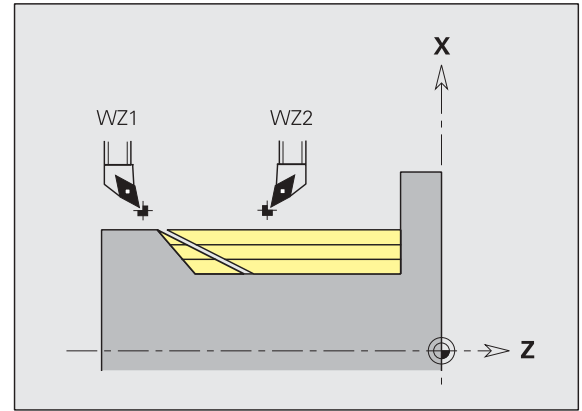
Depending on the contour and the available tools, the machining operation is executed as follows:

- Complete hollowing with one tool. If more than one tool is available, the tool with standard machining direction is used.
- If the final contour element of the area to be hollowed is a transverse element, the tool first cuts towards the transverse element (see figure).
- If the two tools have different clearance angles, the tool with the larger clearance angle is used first.
- If both tools have the same clearance angle, machining starts from the side with the smallest "inward copying angle."



Danger of collision!

During hollowing operations on the inside of contours, the plunging depth of the tool **is not checked**. Select suitable tools.

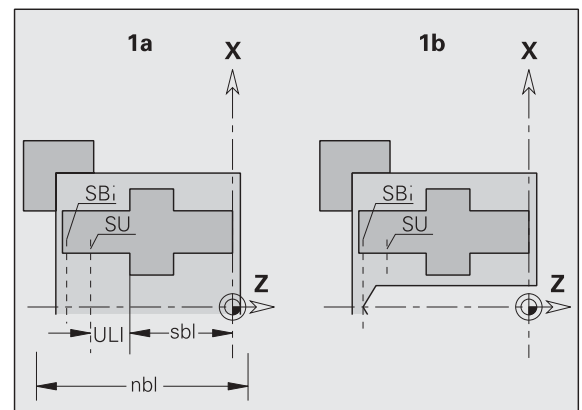


6.16.5 Inside Contours

TURN PLUS machines continuous inside contours up to the transition from the "deepest point" to a greater diameter. In addition, TURN PLUS accounts for

- Cutting limitation, inside
- Overhang length, inside (ULI, machining parameter 4)

when determining the end position for drilling, roughing and finishing operations. TURN PLUS assumes that the usable tool length is sufficient for the respective machining operation; if not, the inside contour is machined according to the tool parameter.



Limits for internal machining operations

■ Predrilling

SBI limits the drilling operation.

■ Roughing

SU or **SU** limit the roughing operation.

SU = basic length of roughing cut (sbl) + overhang length, inside (ULI)

To avoid "residual rings" during the machining process, TURN PLUS leaves an area of 5° in front of the roughing limit.

■ Finishing

sbl limits the finishing operation.

The illustrations show the dimensions (a), the drilling operation (b), the roughing operation (c) and the finishing operation (d).

Example 1

The roughing limit (SU) is **in front of** the inside cutting limit (SBI).

Example 2

The roughing limit (SU) is **behind** the inside cutting limit (SBI).

Abbreviations

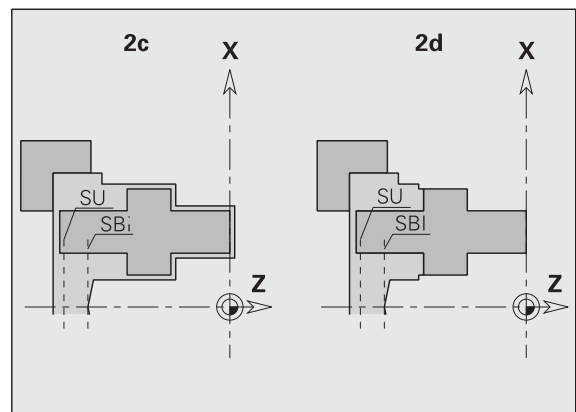
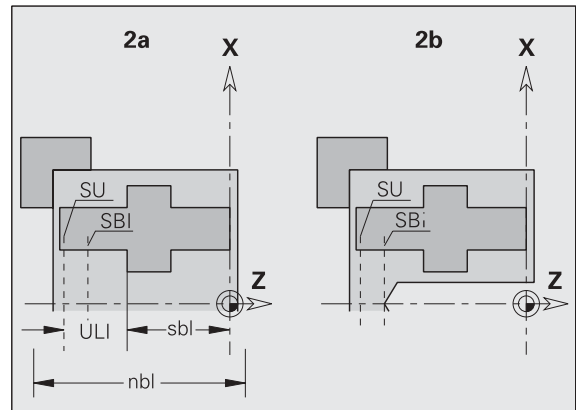
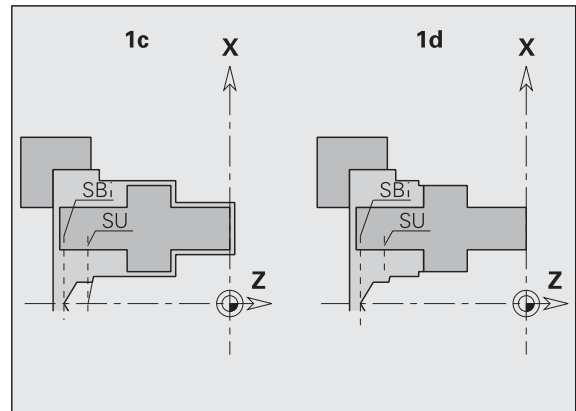
SBI: Cutting limitation, inside

SU: Roughing limitation (SU = sbl + ULI)

sbl: Basic length of roughing cut ("deepest" point of inside contour)

ULI: Overhang length, inside (machining parameter 4)

nbl: Usable tool length (tool parameter)



6.16.6 Drilling

Drilling without definition of fits

TURN PLUS selects tools that permit machining to finished dimensions. First it searches for twist drills, then for boring bars with indexable inserts.

Drilling with definition of fits

TURN PLUS machines the bore hole in two steps.

- Drilling with smaller diameter than the nominal diameter of the hole.
- "Reaming" to finished dimension

6.16.7 Full-Surface Machining

You describe the geometry of the blank and finished part, and TURN PLUS generates the working plan for the **complete workpiece**.

Depending on the machining sequence, after machining the front side TURN PLUS activates an **expert program** for rechucking (machining parameter 21):

- "Rechuck – Full-surface machining": the opposing spindle takes over the workpiece (entry of "UP-UMKOMPL")
- "Parting – Full-surface machining": Bar machining – the workpiece is parted and taken over by the opposing spindle (entry of "UP-UMKOMPLA")

The NC program generated includes front and back machining (including the drilling, milling and inside machining), calling the expert program, and the clamping information of both setups (see also: "4.18.3 Full-Surface Machining")

Preconditions for full-surface machining

- **Program header:** Assignment of spindle to slide for the 2nd setup (input fields: "2nd setup of spindle .. With slides ..").

- **Machining sequence:** Entry "Main machining" RECHUCK or PART after machining the front side (see "6.13.2 Machining Sequence").

For the back machining you can:

- Enter the machining operations after RECHUCK/ PARTING
- Use the same machining sequence as for front side machining (no further entries after RECHUCK/ PARTING).

Continued ►



TURN PLUS only evaluates the information "with/without fit." The type of fit (H6, H7, ..) is of no significance.

Note on machining the rear face

For contours on the rear side (C/Y axis machining) remember the orientation of the XK or X axis and the orientation of the C axis.

Designations:

- Front side: The side toward the working space
- Rear side ("R"): The side away from the working space

These designations also apply to workpieces clamped at the opposing spindle, or to workpieces rechucked for rear-side machining in lathes with one spindle.

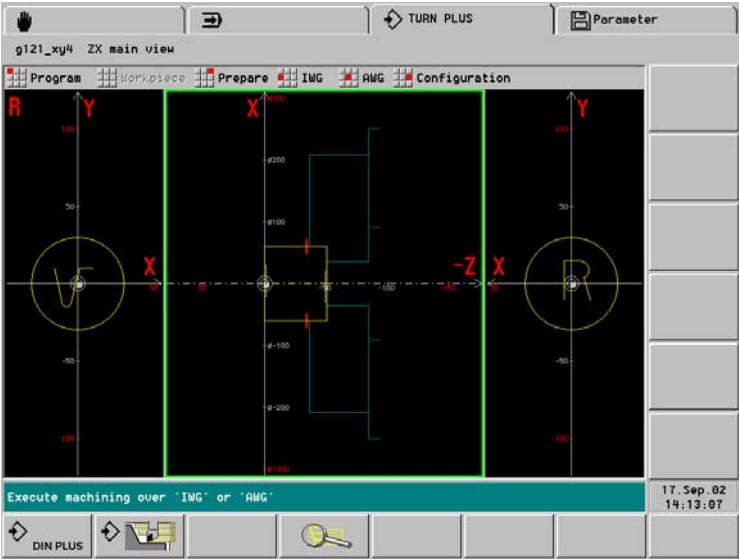
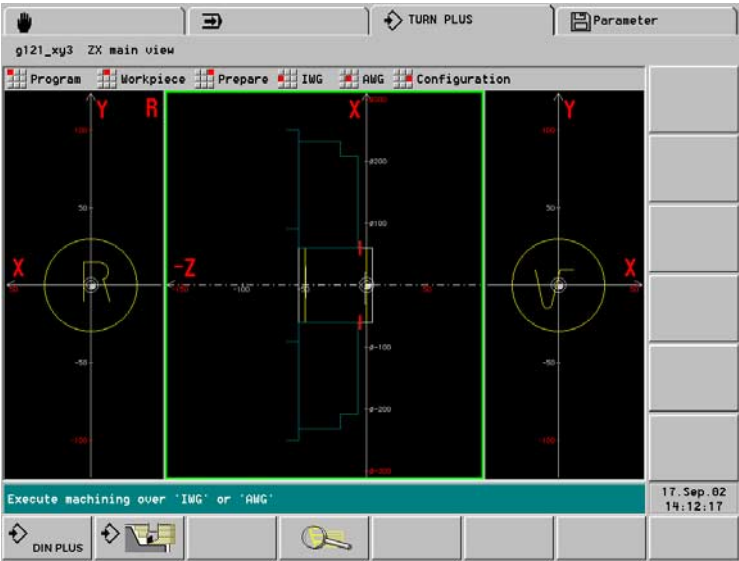


Illustration of lathe with opposing spindle

6.16.9 Shaft Machining

For shafts, TURN PLUS supports rear-face machining of outside contours in addition to standard machining processes. This enables shafts to be completely machined using one setup.

TURN PLUS does **not** support retracting the tailstock and does not check the setup used.

Precondition for shaft machining: The workpiece is clamped at spindle and tailstock.



Danger of collision!

TURN PLUS does **not** monitor for collisions during transverse machining or machining operations on the end face.

Separation point (TR)

The workpiece is divided into **front** and **rear** area. If no separation point has been specified, TURN PLUS sets a separation point at the transition from the largest to a smaller diameter. Position the separation points on outside corners.

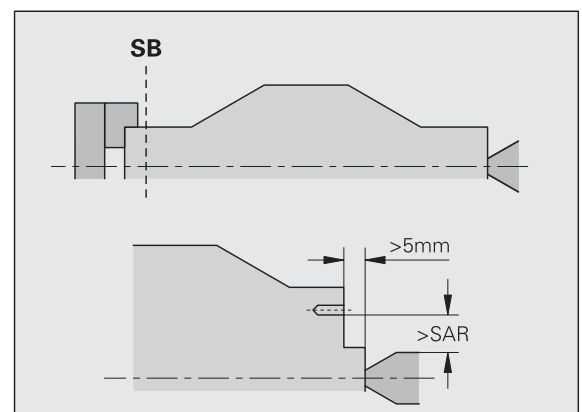
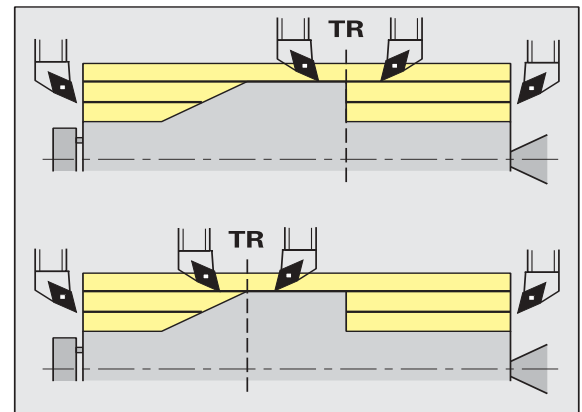
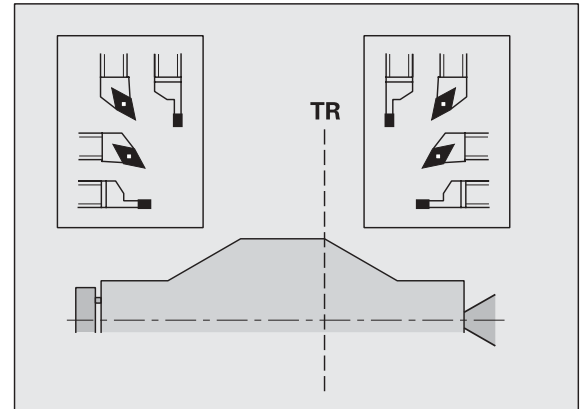
Tools for machining the

- Area on front side: Main machining direction – Z; or primarily “left” recessing or tapping tools, etc.
- Area on rear side: Main machining direction + Z; or primarily “right” recessing or tapping tools, etc.

Setting/editing a separation point: see “6.9.5 “Separation Points”

Protective zones for drilling and milling operations

- TURN PLUS machines drilling and milling contours on the end face (front and rear face) provided that the following conditions are fulfilled:
 - The (horizontal) distance from the end face must be greater than 5 mm, or
 - The distance between chucking equipment and drilling/milling contour must be greater than SAR (SAR: see machining parameter 2).
- If jaws are used for clamping the shaft at the spindle, TURN PLUS accounts for the cutting limitation (SB).



Continued ►

Machining information

■ Chucking the workpiece at the spindle

Ensure that the area, where the blank part is chucked, is premachined. Otherwise, the cutting limitation might adversely affect the machining strategies.

■ Machining of bars

TURN PLUS **does not control** the bar loader and does not move the tailstock and steady rest components. TURN PLUS does not support workpiece adjustment between collet and dead center during machining operations.

■ Machining operations on shaft ends

- Note that the machining sequence defined applies to the entire workpiece, including machining operations on shaft ends.
- The AWG does not machine inside contours on rear ends. If jaws are used for clamping the shaft at the spindle, the rear face is not machined.

■ Longitudinal machining

First the front area is machined, then the rear area.

■ Avoiding collisions

Collisions during machining operations can be avoided as follows:

- Add the retraction of the tailstock, the positioning of the collet, etc., to the DIN PLUS program.
- Add a cutting limitation to the DIN PLUS program.
- Prevent the automatic execution of machining operations by assigning the "Do not machine" attribute or by specifying the machining location in the machining sequence.
- Define for the blank part: Oversize = 0. As a consequence, the front area is not machined (e.g. shafts cut to length and centered shafts).

6.17 Example

On the basis of the production drawing, the working steps for defining the contour of the blank and finished part, the setup procedures and automatic working plan generation are explained.

Creating a program

Select "Program - New."

"New program" dialog box:

- Enter a new program name
- Material – select the material from the fixed-word list
- Press the "program head" button

"Program head" dialog box:

- Enter the spindle and slides for the 1st setup.
- Fill in further input fields as required

Return to "New program" dialog box

"OK" – the new program is ready

Defining a blank

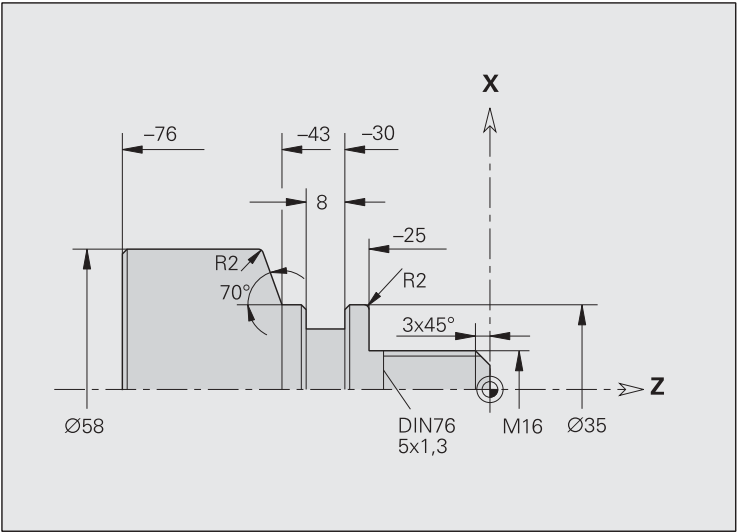
Select "Workpiece - Blank."

Select "Bar."

"Bar" dialog box:

- Diameter = 60 mm
- Length = 80 mm
- Oversize = 2 mm
- "OK" –TURN PLUS depicts the workpiece blank

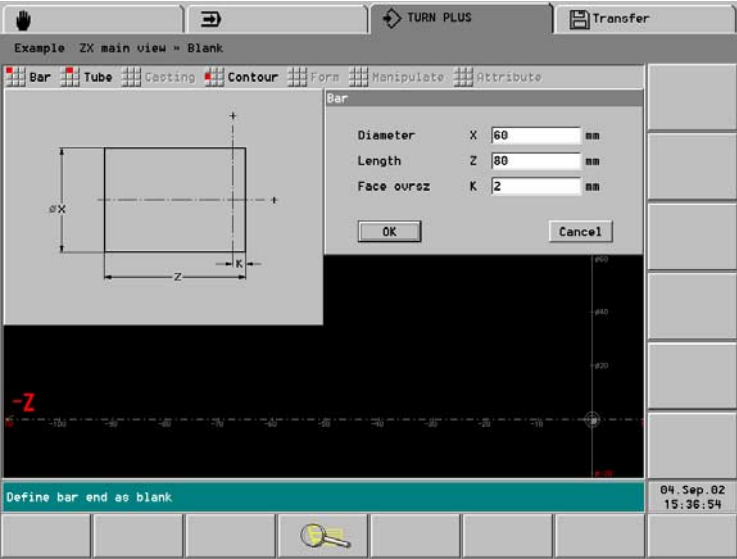
To return to the main menu, press the ESC key.



Undefined chamfers: 1x45°

Undefined radii: 1mm

Workpiece blank: -60 X 80; Material: Ck 45



Defining the basic contour

Select "Workpiece - Finished Part."

"Point" dialog box (starting point of contour):

- X = 0
- Z = 0
- "OK" –TURN PLUS depicts the starting point



Enter
X = 16; Press OK.



Enter
Z = -25; Press OK.



Enter
X = 35; Press OK.



Enter
Z = -43; Press OK.



Enter
X = 58
W = 70 – Press OK



Enter
Z = -76 – press OK

- 2 * Esc key
- "Close contour?" – confirm with "yes" – the basic contour is defined

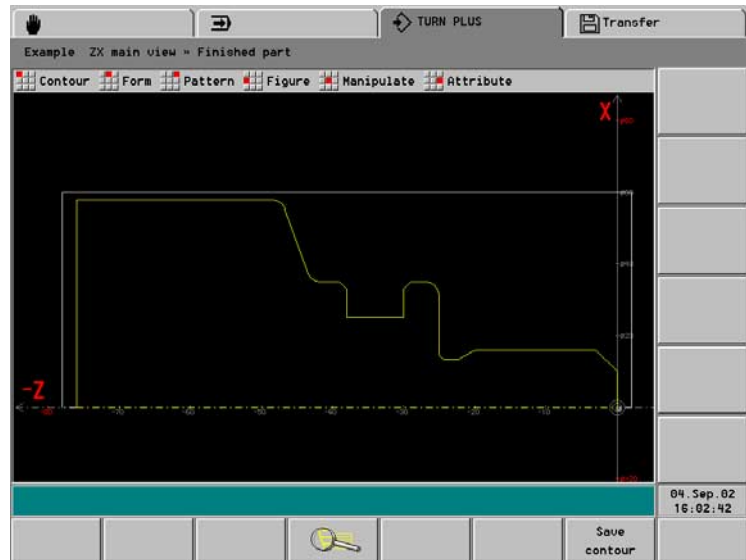
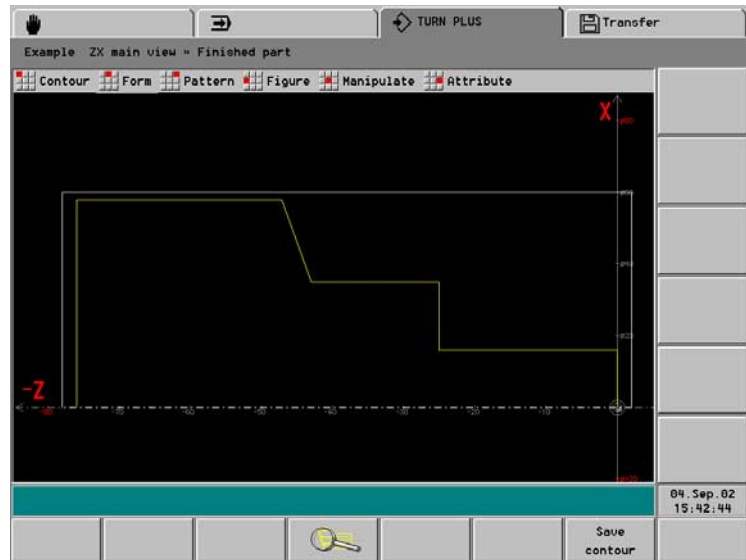
Defining form elements

Select "Form - Chamfer."

- Select "corner threaded stud"
- "Chamfer" dialog box:
 - Chamfer width = 3 mm

Select "Form - Rounding."

- Select "corner for rounding"
- "Rounding" dialog box
 - Rounding radius = 2 mm



Define form elements (continued)

Select "Form - Undercut - Undercut type G."

- Select "Corner for undercut"
- Dialog box "Undercut form G":
 - Undercut length = 5 mm
 - Undercut depth = 1,3 mm
 - Approach angle = 30 °

Select "Form - Recess - Recess type D."

- Select "Basic element for recess"
- "Recess form D" dialog box:
 - Reference point (Z) = -30 mm
 - Undercut width (Ki) = -8 mm
 - Undercut diameter = 25 mm
 - Corners (B): Chamfers; 1 mm

Select "Form -Thread."

- Select "basic element for thread"
- "Thread" dialog box:
 - Select "metric ISO thread"

To return to the main menu, press the ESC key.

Prepare – Chucking

Select "Prepare - Chucking - Chuck."

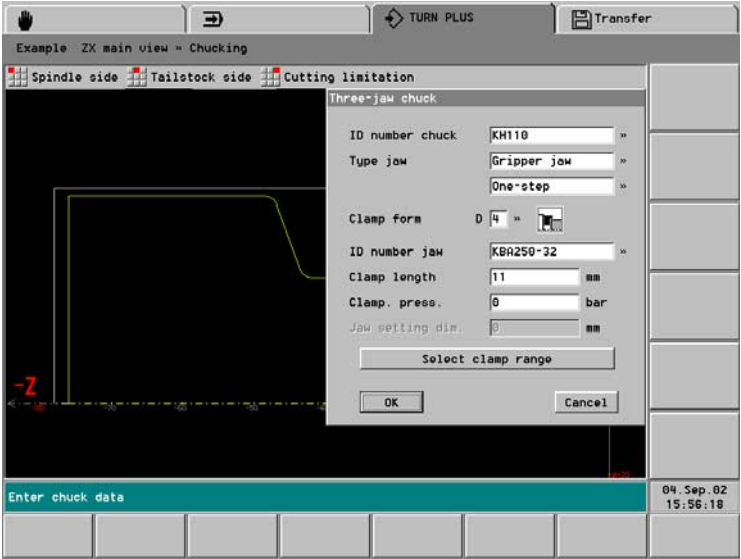
Select "Spindle side -Three-jaw chuck."

"Three-jaw chuck" dialog box:

- Select "ID number of chuck"
- Enter "type of jaw"
- Enter the "clamp form"
- Select the "ID number of jaw"
- Check/enter the "clamp length" and "clamp pressure"
- Define the clamp range (select a contour element that is touched by clamping jaws)

Close the "three-jaw chuck" dialog box –TURN PLUS now depicts the chucking equipment and the cutting limits

To return to the main menu, press the ESC key.



Generating a working plan "blockwise"

Select "AWG - Block-by-block."

TURN PLUS runs a graphic simulation of the machining process block by block.

Select "Accept (working) block."

After the working plan has been generated, select "Accept working plan."

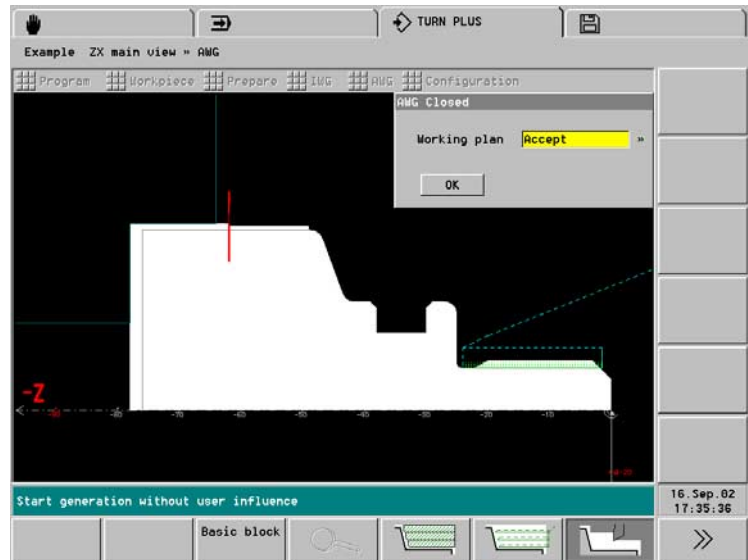
Saving a program

Select "Program - Saving - Complete."

Check the file name; Press OK.

TURN PLUS saves

- The working plan, the contour of the blank and finished part (in one file)
- The NC program (DIN PLUS format)



The AWG generates the working blocks according to the machining sequence and the settings in the machining parameters (see "6.13.2 Machining Sequence and 7.5 Machining Parameters").

meter - Auswahl
Akt. Para Param.-Listen
Auswahl der Maschinendaten-Para
Nr Inhalt der Parameter
1 Maschinenkonfiguration
2 Aggregate der Maschine
3 Allgemeine Achskonfiguration
4 Allgemeine Spindelkonfiguration
5 Aggregatgruppenzuordnung / E
Werkzeugmessen
Maschinenmaße
Revolverbelegungstabelle
Verkettung Multi-WZ
alternativ WZ-Kette
zeige Einstellung
uerungskonfigurierung
k-ahead

0.002

001

001

T

1

X/ 0
Z/ 0.

123456
SDF
C--

AG1 Zy RAHFIE
AG2



7

Parameters

7.1 Parameter Mode of Operation

7.1.1 Parameters

The parameters of the CNC PILOT are grouped as follows:

- **Machine parameters**
These are used to adapt the control to the requirements of the machine, e.g. parameters for components, assemblies, the assignment of axes, slides and spindles.
- **Control parameters**
These are used to configure the control system (machine display, interfaces, measuring system used, etc.).
- **Setup parameters**
These define certain settings required for the production of a specific part (workpiece datum, tool change position, compensation values, etc.).
- **PLC parameters**
These parameters are defined by the machine manufacturer (see Machine Manual).
- **Machining parameters**
These define the strategy for machining cycles and for TURN PLUS.

In this operating mode, the following operating resource and technology parameters are managed in addition (see Chapter "8 Operating Resources":

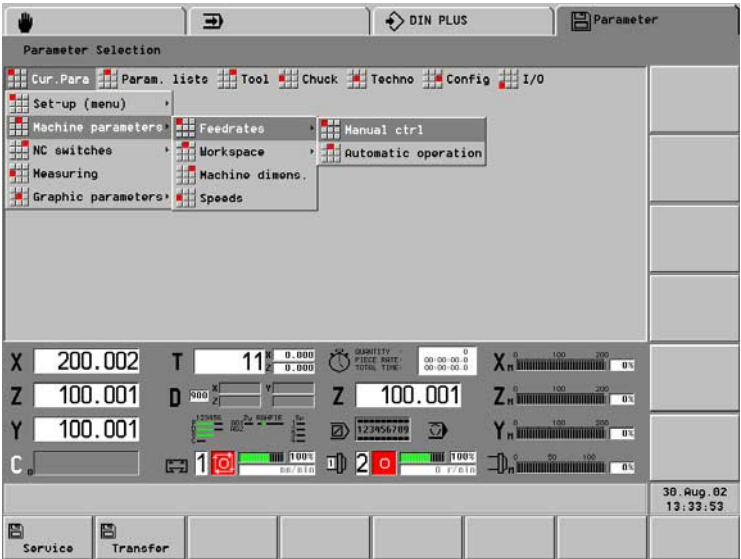
- **Tool parameters**
- **Chucking equipment parameters**
- **Technology parameters (cutting values)**

This Manual describes parameters that can be changed by the user ("system manager" user class). Other parameters are explained in the Technical Manual.








Data transfer and data backup

The CNC PILOT supports **data exchange** of parameters and associated fixed-word lists. All parameters are included for **data backup**.

Data transfer and data backup occur in the Transfer mode of operation – see "10.4 Parameters and Operating Resources."



Main menu of parameter mode

	Current Parameters - frequently used parameters – selectable via menu
	Param.eterlists of the PLC group, setup and machining
	Tool parameters Description of the tools – see "8.1 Tool Database"
	Chuck parameters Description of the chucking equipment – see "8.2 Chucking Equipment Database"
	Technology parameter – see "8.3 Technology database (cutting values)"
	Configuration – contains all parameters (can only be accessed by system manager)
	Input/Output and parameter backup

7.1.2 Editing Parameters

Active Parameters

An overview of frequently used parameters are presented in the "Cur.(rent) para(meter)" drop-down menu so that you can select them without having to know the parameter number.

Editing parameters

If required, log on as system manager (Service mode)

Select "Cur. Para" (Parameter mode)

Select a parameter via the menu - the CNC PILOT opens a window for editing the parameter.

Make the changes

Parameter lists

The parameter groups

- Setup parameters
- Machining parameters
- PLC parameters

are available in the sub-items of "Param. lists." You can select these parameters without being logged on as system manager.

Editing set-up/machining parameters

Select "Param. lists" (Parameter mode)

Select parameter groups

- Setup parameters
- Machining parameters
- PLC parameter

Select parameters

ENTER – the CNC PILOT presents the parameters for editing

Make the changes



Editing configuration parameters

You edit parameters of the "machine" and "control" groups as follows:

Editing parameters

Log on as system manager (Service mode)

Select "config" (Parameter mode)

Parameter number unknown:

Select parameter group (machine, control)

Select parameter ("arrow up/down" or touch pad)

ENTER – the CNC PILOT presents the parameter for editing

Parameter number is known:

"Machine direct / Control direct"

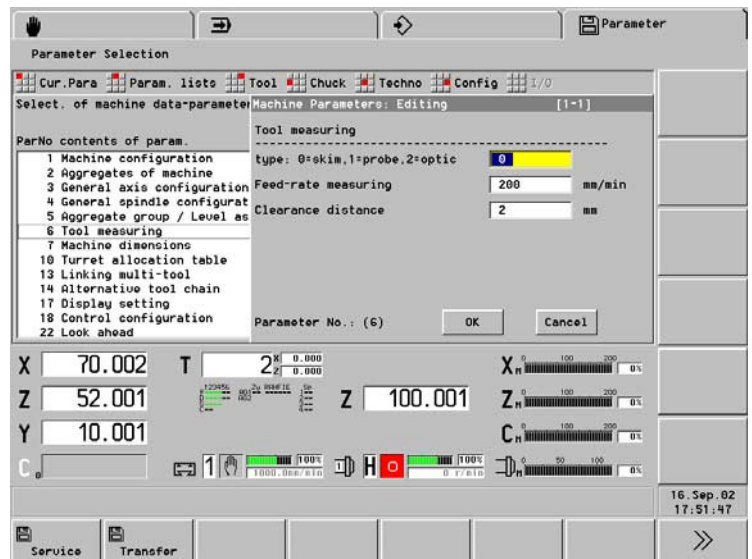
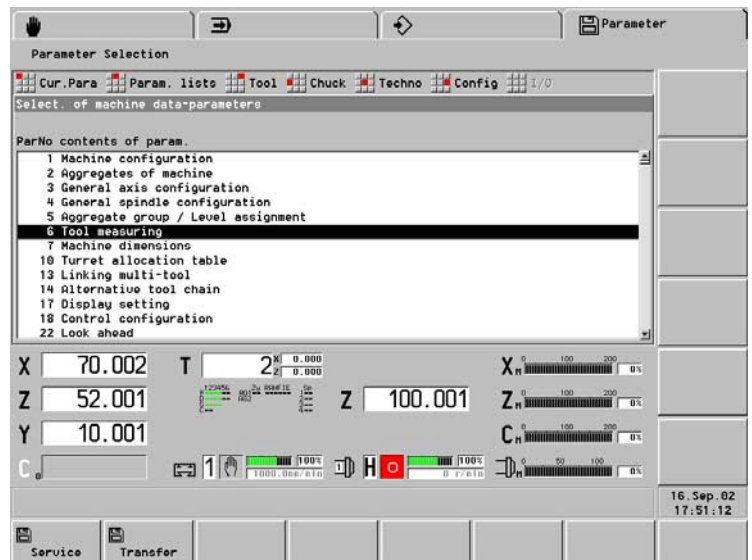
Enter the parameter number

Make the changes

In the submenu of "Config" you can select in addition the parameter groups

- Setup parameters
- Machining parameters
- PLC parameters

The procedure is identical with that described under parameters lists.



- The CNC PILOT checks whether the user is authorized to change a parameter. Log on as system manager if you wish to edit protected parameters. Otherwise, you are only authorized to read parameters.
- Parameters that influence the production of a workpiece cannot be edited in automatic mode.
- Parameters that you as machine operator cannot change are explained in the Technical Manual.

7.2 Machine Parameters

Value ranges of machine parameters

- 1..200: General machine configuration
- 201..500: Slides 1 to 6: 50 positions per slide (NC channel)
- 501..800: Tool carriers 1 to 6: 50 positions per tool carrier
- 801..1000: Spindles 1 to 4: 50 positions per spindle
- 1001..1100: C-axes 1 to 2: 50 positions per C-axis
- 1101..2000: Axes 1 to 16: 50 positions per axis
- 2001..2100: Various machine components

General machine parameters

6 – Tool measuring

These parameters define how to determine the tool lengths in set-up mode.

- **Method** (of tool measuring):
 - 0: Scratch
 - 1: Probe
 - 2: Optical measuring system
- **Measuring feed:** Feed rate for approaching the touch probe
- **Clearance distance:** Minimum distance, which has to be traversed in the opposite direction to the measuring direction, to retract the touch probe after the stylus has deflected.

7 – Machine dimensions

Within the framework of variable programming, machine dimensions can be used in NC programs. The contents and evaluation of machine dimensions depend on the NC program only.

- **Dimension n X, Y, Z, U, V, W, A, B, C (n: 1 to 9)**

17 – Display setting

The "Display mode" defines the position display (actual value display) within the machine display.

- **Actual-value display**
 - 0: Actual value
 - 1: Lag error
 - 2: Distance to be covered
 - 3: Tool tip referenced to machine zero point
 - 4: Slide position
 - 5: Distance between reference cams and zero point
 - 6: Nominal position value
 - 7: Distance between tool tip and slide position
 - 8: IPO target position

General machine parameters (continued)

18 Control configuration

- **PLC takes over workpiece counting**
 - 0: CNC takes over workpiece counting
 - 1: PLC takes over workpiece counting
- **M0/M1 for all NC channels**
 - 0: M0/M1 initiates a STOP on programmed channels.
 - 1: M0/M1 initiates a STOP on all channels.
- **Stop interpreter during tool change**
 - 0: Not active.
 - 1: Active; look-ahead block interpretation is interrupted and only resumed after execution of the T command.

Slide parameters

204, 254, ... Feed rates

Rapid traverse speed and feed rate when slide is moved with the axis-direction keys (jog keys).

- **Rapid traverse contouring speed for manual control**
- **Feed rate contouring speed for manual control**

205, 255, ... Protective-zone monitoring

The protection zone dimensions are defined for individual axes (machine parameter 1116, ...). Define in this parameter whether the protection zone dimensions are to be monitored.

- **Monitoring**
 - 0: Protection zone monitoring off
 - 1: Protection zone monitoring on
- The other parameters are not used at present.

208, 258, ... Thread cutting

If the "coupling/uncoupling path" is **not** programmed in an NC program, parameter values are used.

- **Coupling path:** Acceleration path at the beginning of a threading cut in order to synchronize the feed axis with the spindle.
- **Decoupling path:** Deceleration path at the end of a threading cut.

209, 259, ... Slide deactivation

- **Slide**
 - 0: Deactivate slide.
 - 1: Do not deactivate slide.

Slide parameters (continued)

211, 261, ... Position of probe or optical measuring system

To define the **position of the probe**, enter its external coordinates.

To define the position of the **optical measuring system**, enter the position of the cross hairs (+X/+Z).

Reference: Machine zero point

- **Position of probe/optical measuring system +X**
- **Position of probe –X**
- **Position of probe/optical measuring system +Z**
- **Position of probe –Z**

511..542, 561..592, ... Description of tool holders

Positions of the tool holders relative to the tool-carrier reference point.

- **Distance to carrier reference point X/Z/Y:** Distance between reference point of tool carrier and reference point of tool holder.
- **Compensation X/Z/Y:** Compensation value for the distance between reference point of tool carrier and reference point of tool holder.

Spindle parameters

804, 854, ... Protection-zone monitoring for spindle – not used at present

805, 855, ... General spindle parameters

- **Zero point shift (M19):** Defines the distance between the reference point of the spindle and the reference point of the encoder. After receiving the reference pulse from the encoder, the value determined is transferred.
- **Number of revolutions for backing off:** Number of spindle revolutions after the spindle has stopped in Automatic mode. (When using low spindle speeds, additional spindle revolutions are necessary to relieve the tool.)

806, 856, ... Tolerance values, spindles

- **Speed tolerance [%]:** The system proceeds from a G0 block to a G1 block as soon as the status "Speed reached" is attained. When the speed lies within the tolerance range, the status is attained. The tolerance value refers to the nominal speed.
- **Position window, position [°]:** After the spindle has stopped at a defined position (M19), the system proceeds to the next block as soon as the status "Position reached" is attained. When the difference between nominal and actual value lies within the tolerance range, the status is attained. The tolerance value refers to the nominal speed.
- **Speed tolerance, synchronization [rpm]:** Criterion for status "Synchronization reached."
- **Position tolerance, synchronization [°]:** Criterion for status "Synchronization reached."

The parameter settings for the slave spindle are the decisive ones.

Continued ►

Parameters for spindles (continued)

Status "Synchronization reached": If the actual speed difference and the actual position difference between the spindles to be synchronized lies within the tolerance range, the status is reached. When the status "Synchronization reached" is attained, the torque of the guided spindle is limited.

Note: The tolerances which are actually attainable on the machine must not be undercut when setting these parameters. The value programmed here must be set to a value greater than the sum of the maximum differences in the speeds of the guiding and guided spindles (approx. 5 to 10 rpm).

807, 857, ... Angular offset measuring (G906), spindles

- Evaluation:** G906 Measuring angular offset during spindle synchronization
- **Maximum permissible change in position:** Tolerance window for the change of position offset after the spindles have gripped the workpiece at both ends during synchronized operation. If the offset change exceeds the maximum value, an error message appears. A normal fluctuation of approx. 0.5° must be taken into account.
 - **Waiting time, measuring offset:** Measuring period (duration)

808, 858, ... Cut-off control (G991), spindle

After the parting operation has been completed, the phase angle of the two synchronized spindles changes without the nominal values (speed/angle of rotation) being changed. If the speed difference is exceeded during the monitoring time, the workpiece is considered as being cut off.

- Evaluation:** G991 Controlled parting using spindle monitoring
- **Speed difference**
 - **Monitoring time**

809, 859, ... Load monitoring, spindles

- Evaluation:** Load monitoring
- **Start-up time for monitoring [0..1000 ms]** – (only evaluated if "Omit paths of rapid traverse" is active) The load monitoring function is not activated if the nominal acceleration of the spindle exceeds the limit value (limit value = 15% of acceleration ramp / brake ramp). As soon as nominal acceleration falls below the limit value, the monitoring function is activated after the start-up time for monitoring has elapsed.
 - **Number of measured values to be averaged [1..50]:** The mean value is calculated from the number of values to be averaged. This reduces the sensitivity to short-term peak load during monitoring.
 - **Reaction delay time P1, P2 [0..1000 ms]**
The system reports a limit violation as soon as the delay time for P1 or P2 (limit torque 1 or 2) has been exceeded.
 - **Maximum torque** – not used at present

Parameters for C axes

1007, 1057 Backlash compensation, C axis

In backlash compensation the “value of backlash compensation” is calculated into every change in direction.

■ Type of backlash compensation

- 0: No backlash compensation
- 1: Encoder is built into the motor. The backlash compensation accounts for the reversal error between the motor and table. During each change in direction, the nominal value is adjusted by the value entered in the “backlash compensation value” option.
- 2: With direct measurement, the backlash compensation compensates the reversal error between the motor and encoder. During each change in direction, the nominal value is corrected by the value entered in the “Backlash compensation value” option.

■ Backlash compensation value:

- For type=1: compensation value with positive sign
- For type=2: compensation value with negative sign

1010, 1060 Load monitoring, C axis

Evaluation: Load monitoring

- **Start-up time for monitoring [0 to 1000 ms]**—(evaluated if “Omit paths of rapid traverse” is active): The load monitoring function is not activated if the nominal acceleration of the spindle exceeds the limit value (limit value = 15% of acceleration ramp / brake ramp). As soon as nominal acceleration falls below the limit value, the monitoring function is activated after the start-up time for monitoring has elapsed.
- **Number of measured values to be averaged [1 to 50]**: The mean value is calculated from the number of values to be averaged. This reduces the sensitivity to short-term peak load during monitoring.
- **Maximum torque**—not used at present
- **Reaction delay time P1, P2 [0 to 1000 ms]**
The system reports a limit violation as soon as the delay time for P1 or P2 (limit torque 1 or 2) has been exceeded.

1016, 1066 Limit switches and rapid traverse rates, C axis

- **Rapid traverse rate, C axis**: Maximum speed for spindle positioning

1019, 1069 General data, C axis

This parameter is evaluated if “pre-positioning” is switched on (“configuration code 1”—machine parameter 18). For digital drives, pre-positioning is usually not necessary.

- **Pre-positioning of spindle with M14**: Angle at which the spindle is positioned before the C axis is swiveled in.

Parameters, C-axis (continued)

1020, 1070 Angle compensation in C axis – Parameters are entered by the machine tool builder.

1021..1026, 1071..1076 Compensation values of C axis – Parameters are entered by the machine tool builder.

Parameters for linear axes

1107, 1157, ... Backlash compensation, linear axes

In backlash compensation the "value of backlash compensation" is calculated into every change in direction.

- **Type of backlash compensation**
 - 0: No backlash compensation
 - 1: Encoder is built into the motor. The backlash compensation accounts for the reversal error between the motor and table. During each change in direction, the nominal value is adjusted by the value entered in the "backlash compensation value" option.
 - 2: With direct measurement, the backlash compensation compensates the reversal error between the motor and encoder. During each change in direction, the nominal value is corrected by the value entered in the "Backlash compensation value" option.
- **Backlash compensation value:**
 - For type=1: compensation value with positive sign
 - For type=2: compensation value with negative sign

1110, 1160, ... Load monitoring, linear axis

Evaluation: Load monitoring

- **Start-up time for monitoring [0..1000 ms]** – (evaluated if "Omit paths of rapid traverse" is active) The load monitoring function is not activated if the nominal acceleration of the spindle exceeds the limit value (limit value = 15% of acceleration ramp / brake ramp). As soon as nominal acceleration falls below the limit value, the monitoring function is activated after the start-up time for monitoring has elapsed.
- **Number of measured values to be averaged [1..50]:** The mean value is calculated from the number of values to be averaged. This reduces the sensitivity to short-term peak load during monitoring.
- **Maximum torque** – not used at present
- **Reaction delay time P1, P2 [0..1000 ms]**

The system reports a limit violation as soon as the delay time for P1 or P2 (limit torque 1 or 2) has been exceeded.

Continued ►

Parameters for linear axes (continued)

1112, 1162, ... Traverse to fixed stop (G916), linear axis

These parameters apply to the linear axis for which G916 has been programmed.

Evaluation: G916 Traverse to fixed stop

- **Lag error limit:** The slide is stopped as soon as the lag distance (difference between actual and nominal position) has reached the lag error limit.
- **Reversing path:** After a fixed stop has been reached, the slide is reversed by the reversing path (stress relief).

1114, 1164, ... Zero offset when converting linear axes

- **NC zero offset:** Length by which the machine zero point is shifted during conversion (G30).

1115, 1165, ... Controlled parting (G917), linear axis

These parameters apply to the linear axis for which G917 has been programmed.

Evaluation: G917 Controlled parting using lag error monitoring

- **Lag error limit:** The slide is stopped as soon as the difference between actual and nominal position has reached the lag error limit. As a result, the CNC PILOT generates the message "Lag error detected."
- **Feed rate** when moving the linear axis using lag error monitoring.

1116, 1166, ... Limit switches, protection zone, linear axis feeds

- **Protection zone dimension, negative**
- **Protection zone dimension, positive**
Dimensions for protective-zone monitoring. Reference: Machine zero point
- **Rapid traverse rate** in Automatic mode
- **Reference dimension:** Distance between reference point and machine zero point

1120, 1170, ... Slide offset compensation, linear axis – Parameters are entered by the machine tool builder.

Component parameters

Parameters 2003 ... 2013 are not used at present

7.3 Control Parameters

Control Parameters

1 – Settings

- **Deactivate printer output:** Using the PRINTA command in an NC program, you can output data on a printer (see also control parameter 40, ...).
 - 0: Output deactivated
 - 1: Output activated.
- **Metric / Inch:** Definition of the system of measurement
 - 0: Metric system
 - 1: Inch system
- **Display format** of position display (actual value display)
 - 0: Format 4.3 (4 integer positions, 3 decimal places)
 - 1: Format 3.4 (3 integer positions, 4 decimal places)



- In DIN PLUS programs, the unit of measure is defined in the program head and is independent of the setting made here.
- Restart the CNC PILOT after having changed the unit of measure.

8 – Load monitoring settings

Evaluation: Load monitoring

- **Factor for torque limit value 1**
- **Factor for torque limit value 2**
- **Factor for work limit value**

The CNC PILOT calculates as follows:

Limit value = reference value * factor for limit value

- **Minimum torque [% of rated torque]:**
Reference values that remain below this value are raised to this minimum torque value. This prevents that limit values are exceeded as a result of minor torque differences.
- **Maximum file size [KB]:**
If the data during measured value registration exceed the maximum file size, the "oldest" values are overwritten.
Approximate value: For one component per minute of program run time approximately 12 KB.

10 – Post-process measuring


Evaluation: Post-process measuring

- **Activate measuring function**
 - 0: Postprocess measuring function off
 - 1: Postprocess measuring function on – the CNC PILOT is ready to receive data

Continued ►

Control parameters (continued)


- **Measuring mode**
 - 1: Post-process measuring
- **Measured-value coupling**
 - 0: New measured values overwrite old measured values.
 - 1: New measured values are not accepted until old measured values are evaluated.

 In control parameters 40, ..., the serial interface is selected and the interface parameters are defined.

11 – FTP parameters

Evaluation: Data transfer using FTP (file transfer protocol)

- **User name:** Name of one's own station
- **Password**
- **Address/name of FTP server:** Address/name of remote station
- **Use FTP**
 - 0: No
 - 1: Yes

 The parameters can also be set using the transfer functions.

20 – Time determination for simulation: general

Non-cutting time for the "time counting" function.

Evaluation: Time counting (Simulation mode)

- **Tool change time [sec]**
- **Gear shifting time [sec]**
- **Time allowance for M functions [sec]:** All M functions are rated with this time. If an M function is also specified in control parameter 21, the time allowances entered here are added.

21 – Time determination for simulation: M function

Individual time allowances for a maximum of 10 M functions.

Evaluation: Time calculation (Simulation mode)

- **1..10. M function** – Number of the respective M function
- **Time allowance [sec]** – individual time allowance. The time calculation function in simulation mode adds the specified time to the time allowance entered in control parameter 20.

Control parameters (continued)

22 – Simulation: Default window size (X, Z)

The simulation function adapts the window size to the workpiece blank. If no blank part is programmed, the CNC PILOT uses a standard window size.

Evaluation: Simulation mode

- **Zero position X** – Distance of the origin of the coordinate system from the lower window frame
- **Zero position Z** – Distance of the origin of the coordinate system from the left window frame
- **Delta X** – Vertical expansion of the graphics window
- **Delta Z** – Horizontal expansion of the graphics window

23 – Simulation: Default blank

If no blank part is programmed, the CNC PILOT uses a standard blank.

Evaluation: Simulation mode

- **Outside diameter**
- **Length of blank part**
- **Right edge of blank part**(Allowance) reference: Workpiece zero point
- **Inside diameter** for hollow cylinders, solid blanks: 0

24 – Simulation: Color table for feed lengths

The feed travel of a tool is displayed in the color assigned to the respective turret location.

Evaluation: Simulation mode

- **Color for turret position n (n: 1..16)** – color code:
 - 0: light green (default color)
 - 1: dark gray
 - 2: light gray
 - 3: dark blue
 - 4: light blue
 - 5: dark green
 - 6: light green
 - 7: dark red
 - 8: light red
 - 9: yellow
 - 10: white

27 – Simulation: Settings

The machining simulation and the control graphics (TURN PLUS) are delayed by the “path delay” time after each path that has been simulated graphically. The simulation speed can thus be influenced. Smallest unit: 10 msec

Evaluation: Simulation mode

- **Path delay (machining)**

Control parameters (continued)


40 – Allocation to interfaces

The interface parameters are saved in the parameters 41 to 47. In parameter 40 the machine tool builder assigns an interface description to an encoder. The Transfer mode of operation uses the parameters of the interface defined under "external input/output."

Meaning of data to be entered:


■ 1..7: Interfaces 1..7, for example, "2 = interface 2" described in control parameter 42.

- External input/output
- DataPilot 90
- Printer
- Post-process measuring
- 2nd keyboard (or card reader)

 The parameter settings are made by the machine-tool builder.


41..47 – Interfaces

The CNC PILOT stores the "settings" of the serial interfaces and the printer interface in these parameters.

 You define the parameter settings in the Transfer mode of operation.

48 – Transfer directory

- NETWORK directory
Path of directory offered and indicated for data transfer with NETWORK.

 You define the parameter settings in the Transfer mode of operation.

196 – SIK number

The CNC PILOT checks whether certain options are available on your system. For configuring additional options, your machine manufacturer must be informed of your board number.

Control parameters (continued)

197 – Option passwords

The options available on your CNC PILOT can be activated for a limited period of time. Enter "9999" in the next free input field and restart the CNC PILOT. All options will now be available for a limited period of time.



Options can be made available a limited number of times and cannot be transferred to other systems.

301 ff – Display type 1..6 manual control/automatic

The machine display comprises 12 fields which can be configured and are arranged as follows:

Field 1	Field 5	Field 9
Field 2	Field 6	Field 10
Field 3	Field 7	Field 11
Field 4	Field 8	Field 12

- **Symbol, field n** (n: 1..12): code number of the "symbol" to be displayed here (for code numbers, see following pages).
- **Slide / Spindle:** Define to which slide, spindle or C-axis the display refers. (The CNC PILOT automatically identifies whether the symbol refers to a slide, spindle or C-axis.)
 - 0: The component that has been selected using the slide/spindle selection key is displayed.
 - >0: Number of slide/spindle/C-axis
- **Component group:** Always enter 0.

Code numbers for "symbols"

0 Special code, no display

1 Actual value in X



2 Actual value in Z



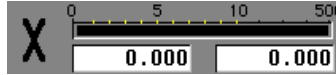
3 Actual value in C



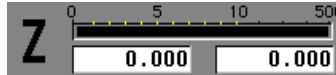
4 Actual value in Y



5 Actual value and distance-to-go in X display



6 Actl value and distance-to-go in Z display



8 Actl value and distance-to-go in Y display



10 All principal axes



11 All secondary axes



12 Actl value in U (auxiliary axis)



13 Actl value in V (auxiliary axis)



14 Actl value in W (auxiliary axis)



Code numbers for "symbols"

15 Actl value in A (auxiliary axis)



16 Actl value in B (auxiliary axis)



17 Actl value in C (auxiliary axis)



21 Tool display with comp. values (DX, DZ)



22 Tool display with identification number



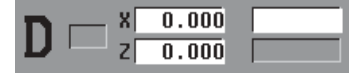
23 Additive compensation



25 Tool display with tool life information



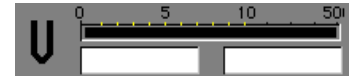
26 Display for multiple tools with compensation values (DX, DZ)



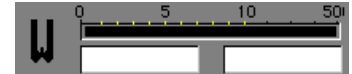
30 Actual value and distance-to-go in U display



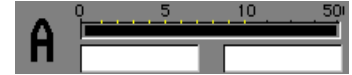
31 Actual value and distance-to-go in V display



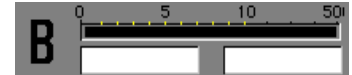
32 Actual value and distance-to-go in W display



33 Actual value and distance-to-go in a display



34 Actual value and distance-to-go in b display



Code numbers for "symbols"	
35	C actual-value and information
41	Quantity information and time per unit
42	Quantity information
43	Time per unit
45	M01 and skip levels
60	Spindle and speed information
61	Actual/nominal speed
69	Actl/noml feed rate
70	Slide and feed rate information
71	Channel display
81	Overview of enables
88	Load display A axis (aux. axis)

89	Load display B axis (aux. axis)
90	Load display C-axis (aux. axis)
91	Load display Spindle
92	Load display X-axis
93	Load display Z-axis
94	Load display C-axis
95	Load display Y-axis
96	Load display U axis (auxiliary axis)
97	Load display V axis (auxiliary axis)
98	Load display W axis (auxiliary axis)
99	Blank field

7.4 Set-Up Parameters



Recommendation: Use **"Cur. Para– Set-up (menu) – ... "** to edit the parameters. In the other menu items the parameters are listed without the axes.

Set-up parameters

Workpiece zero point

For each slide:

- Workpiece zero point, main spindle (reference: machine zero point)
- Workpiece zero point, opposing spindle (reference: machine zero point, opposing spindle)

"PgUp/PgDn" switches to the next/previous slides.

The "Workpiece zero point, counterspindle" is derived from "Machine zero point - Zero offset" (machine parameters 1114, 1164, ..). It is activated with "G30 H1 ..."

- Datum position, main spindle X, Y, Z – Slide 1
- Datum position, main spindle X, Y, Z – Slide 2
- ...
- Datum position, counterspindle X, Y, Z – Slide 1
- Datum position, counterspindle X, Y, Z – Slide 2
- ...



Set the workpiece zero point in the Manual Control mode.

Tool change position

The CNC PILOT manages the tool change point for each slide. "PgUp/PgDn" switches to the next/previous slides.

The tool change position defines the distance to the machine zero point.

- Tool change position X, Y, Z – Slide 1
- Tool change position X, Y, Z – Slide 2
- ...



Set the workpiece zero point in the Manual Control mode.

Set-up parameters (continued)

Datum - Oversize G53/G54/G55

The CNC PILOT manages allowances for zero point offsets for each slide. "PgUp/PgDn" switches to the next/previous slides.

- Oversize X, Y, Z – Slide 1
- Oversize X, Y, Z – Slide 2
- ...

Datum shift C-axis

- Datum shift, C-axis 1
- Datum shift, C-axis 2



- Influences the C-axis actual value.
- The zero point shift G152 is added to this parameter.

Tool life monitoring

- Tool life switch – tool life/quantity monitoring
 - 0: OFF
 - 1: ON
- Load monitoring
 - 0: OFF
 - 1: ON

Additive compensation

The CNC PILOT manages 16 compensation values (for each X and Z). The compensation values can be activated/deactivated in the NC program (see G149, G149 Geo).

- Compensation 901..916 in X
- Compensation 901..916 in Z



If an additive compensation value is changed in Automatic mode, this parameter is changed accordingly.

Deletion level/cycle

A deletion level can be assigned a deletion cycle. As a result, NC blocks containing the specified deletion level are executed each nth time.

- Deletion level [0..9]
- Deletion cycle [0..99]
 - 0: NC blocks with this deletion level are never executed.
 - 1: NC blocks with this deletion level are always executed.
 - 2..99: NC blocks with this deletion level are executed each 2nd to 99th time.



Activate/deactivate the skip levels in automatic mode.

7.5 Machining Parameters

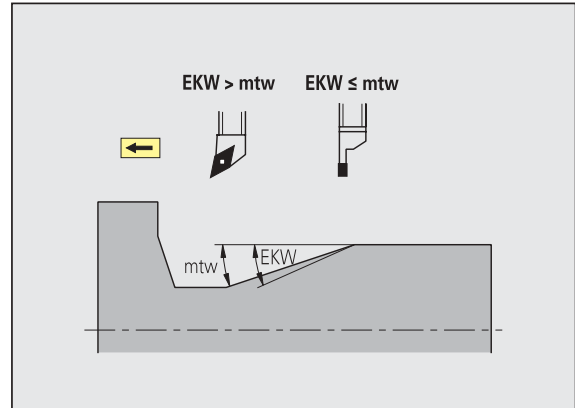


Machining parameters are used by the work plan generation (TURN PLUS) and various machining cycles.

1 – Global finished-part parameters (roughness/limit values)

All elements of the finished part are machined according to the "ORA" and "ORW" parameter values. (Evaluation: Finishing cycle G890).

- **Type of roughness [ORA]** – Type of surface roughness
 - 0: No roughness value
 - 1 – Rt: Peak-to-valley in [μm]
 - 2 – Ra: Mean roughness in [μm]
 - 3 – Rz: Determined peak-to-valley roughness in [μm]
 - 4 – Vr: Direct feed entry in [mm/rev]
- **Roughness values [ORW]:** Roughness or feed rate values
- **Permissible inward copying angle [EKW]:** Limiting angle for recessing contour areas for distinguishing between turning and recessing cycles.
 - $\text{EKW} > \text{mtw}$: Relief turn
 - $\text{EKW} \leq \text{mtw}$: Undefined recess (no form element) (mtw = contour angle).



2 – Global technology parameters

Tool selection, tool change, speed limitation

- **Tool off .. [WD]** – When selecting a tool, TURN PLUS accounts for the:
 - 1: Current turret assignment
 - 2: Current turret assignment in the first place, and the tool database in the second place
 - 3: Tool database
 - **TURN PLUS turret [RNR]** – Precondition "WD=1 or WD=2": RNR determines which turret assignment is used:
 - 0: Current turret assignment in Machine mode
 - 1: TURN PLUS-specific turret assignment (see "6.7.2 Setting Up a Tool List").
 - **Traversing mode to tool change position [WP]** – determines the mode of approach and tool change position. The sequence in which the axes are traversed is defined in the IWG or, for the AWG, in the respective machining parameters.
 - 1: Approaching the tool change position in rapid traverse (G0).
- IwG** – Definition of approach mode and tool change position: "Cycle - Move to tool change point" menu item

Continued ►

- AWG** – Definition of approach mode: Respective machining parameters; Tool change position: Defined tool change point
- 2: Moving to tool change position with G14.
 - 3: Moving to a calculated change position with G0 –TURN PLUS uses the current and subsequent tool to calculate the optimum change position
- **Speed limitation [SMAX]:** Global speed limitation –You can define a lower speed limitation in the TURN PLUS program head (see "6.2.2 Program Head").

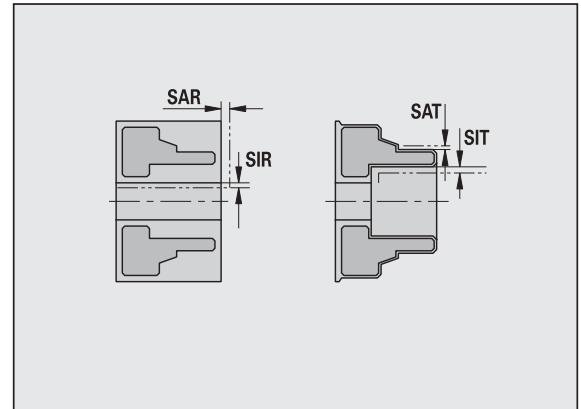
Global safety clearance

- **External on blank [SAR]** – Safety distance on the outside blank
- **Internal on blank [SIR]** – Safety distance on the inside blank
- **External on machined part [SAT]** – Safety distance on the outside premachined workpiece contour
- **Internal on machined part [SIT]** – Safety distance on the inside premachined workpiece contour

TURN PLUS takes account of SAR/SIR for all rough-turning cycles and for centric predrilling.

SAT/SIT on the premachined parts apply to:

- Finish-machining
- Recess turning
- Contour recessing
- Recessing
- Thread cutting
- Measuring

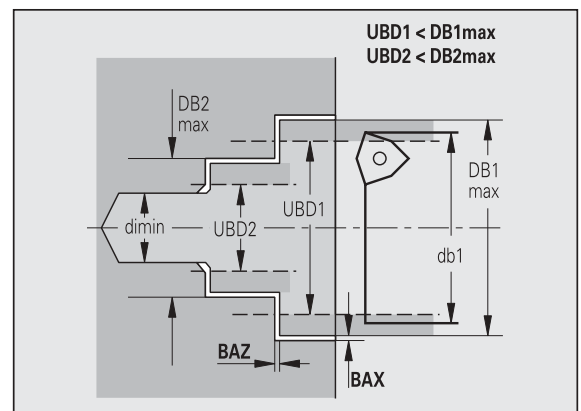


3 – Centric predrilling

Centric predrilling - tool selection, allowances

For predrilling, a maximum of three drilling steps is used:

- 1st predrilling step (diameter limit UBD1)
 - 2nd drilling step (diameter limit UBD2)
 - Finish-drilling step
- **1st drilling diameter limit [UBD1]**
- 1st predrilling step, where $UBD1 < DB1_{max}$
 - Tool selection: $UBD1 \leq db1 \leq DB1_{max}$
- **2nd drilling diameter limit [UBD2]**
- 2nd predrilling step, where $UBD2 < DB2_{max}$
 - Tool selection: $UBD2 \leq db2 \leq DB2_{max}$
- **Finish drilling** is performed with: $dimin \leq UBD2$
- Tool selection: $db = dimin$



Continued ►

Designations:

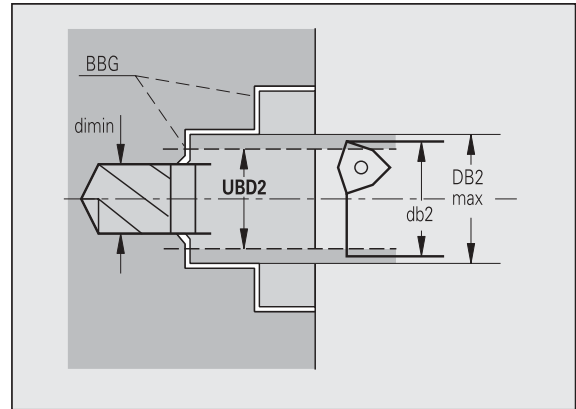
- db1, db2: Drill diameter
- DB1max/DB2max: Maximum inside diameter 1st/2nd drilling step
- dimin: Minimum inside diameter
- BBG – drilling limitation elements: Contour elements intersected by UBD1/UBD2



UBD1/UBD2 have no effect when "Centric predrilling" has been defined as main machining operation followed by "Finish-drilling" as submachining operation in the machining sequence (see "6.12.2 Machining Sequence").

Prerequisite: $UBD1 > UBD2$

UBD2 must permit subsequent inside machining with boring bars.



- **Point angle tolerance [SWT]** – If the drilling limitation element is a diagonal element, TURN PLUS prefers using a twist drill with suitable point angle.

SWT: Permissible point angle tolerance

If no suitable twist drill is available, an indexable-insert drill is selected for the predrilling operation.

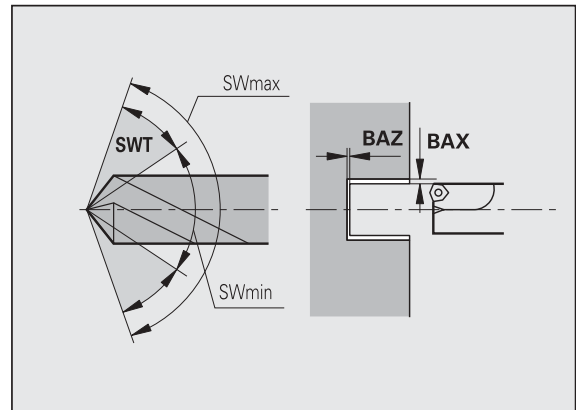
- **Drilling allowance – Diameter [BAX]** – Machining allowance on the drilling diameter (X direction - radius value)

- **Drilling allowance – Depth [BAZ]** – Machining allowance on drilling depth (Z direction)



BAZ is not considered if

- A subsequent internal finishing cycle is not possible because the diameter is too small.
- During finish-machining blind holes, "dimin < 2 * UBD2."



Predrilling - Traverse modes/Safety clearance

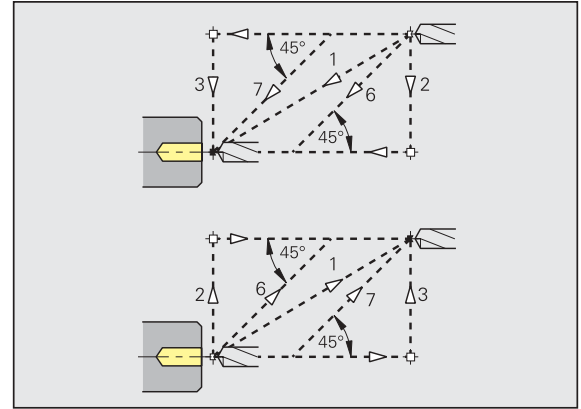
■ Approach for predrilling [ANB]

■ Traverse to tool change point [ABW]

Entries for retracting a tool:

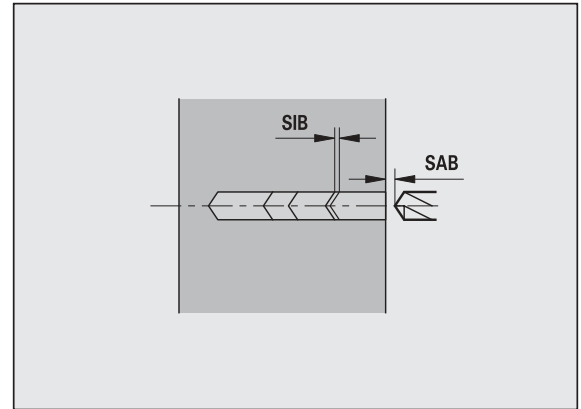
- 1: Simultaneous, X and Z direction
- 2: First X direction, then Z
- 3: First Z direction, then X
- 6: Coupled motion, first X, then Z
- 7: Coupled motion, first Z, then X

Approach and departure are in rapid traverse (G0).



■ Safety clearance to workpiece blank [SAB]

■ Internal safety clearance [SIB] – for deep-hole drilling (retraction distance B for G74).



Predrilling - Machining

■ Drilling depth ratio [BTV] – TURN PLUS checks the 1st and 2nd drilling step. The predrilling step is performed with:

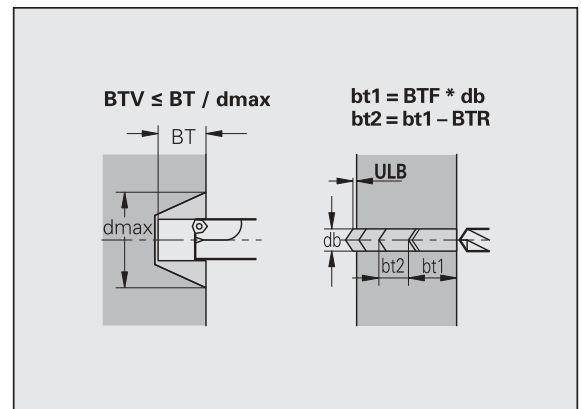
$$BTV \leq BT / d_{max}$$

■ Drilling depth factor [BTF] – 1st drilling depth for deep-hole drilling cycle (G74):

$$bt1 = BTF * db$$

■ Drilling depth reduction [BTR] – Reduction for deep-hole drilling cycle (G74): $bt2 = bt1 - BTR$

■ Overhang length - predrilling [ULB] – Through-drilling length



4 – Roughing

Roughing –Tool and Machining Standards

Finishing tools are defined according to machining location and main machining direction (MMD) via setting angle and point angle. In addition:

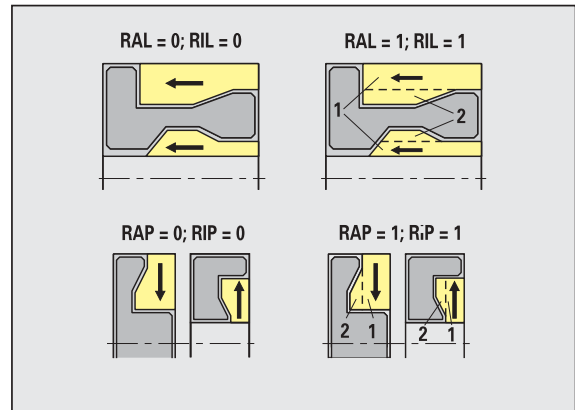
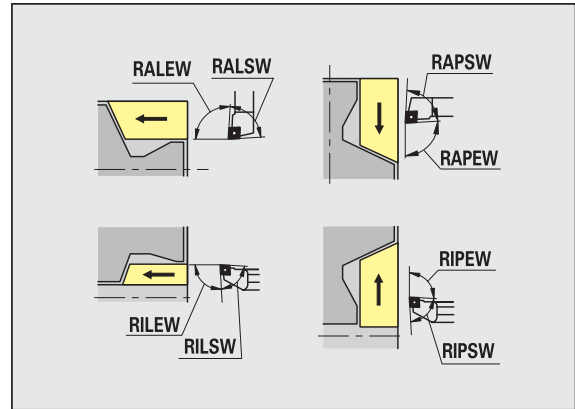
- Use of standard roughing tools is preferred.
- Tools that enable full-surface machining are used as an alternative.
- **Setting angle – external/longitudinal [RALEW]**
- **Point angle – external/longitudinal [RALSW]**
- **Setting angle – external/transverse [RAPEW]**
- **Point angle – external/transverse [RAPSW]**
- **Setting angle – internal/longitudinal [RILEW]**
- **Point angle – internal/longitudinal [RILSW]**
- **Setting angle – internal/transverse [RIPEW]**
- **Point angle – internal/transverse [RIPSW]**

Parameter for machining contour areas:

- **Standard/Complete – external/longitudinal [RAL]**
- **Standard/Complete – internal/longitudinal [RIL]**
- **Standard/Complete – external/transverse [RAP]**
- **Standard/Complete – internal/transverse [RIP]**

Entry:

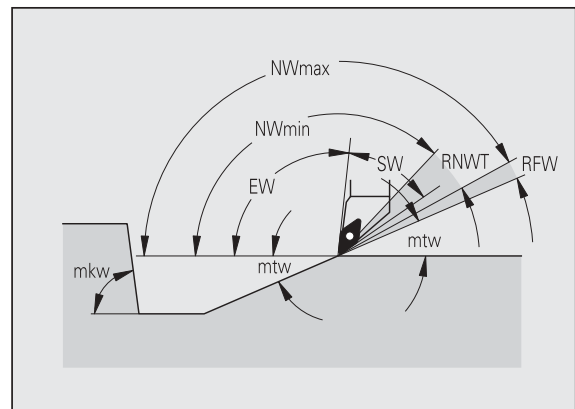
- 0: Complete roughing cycle, including plunge-cutting. TURN PLUS looks for a tool for full-surface machining.
- 1: Standard roughing cycle without plunge-cutting



Roughing –Tool tolerances and allowances

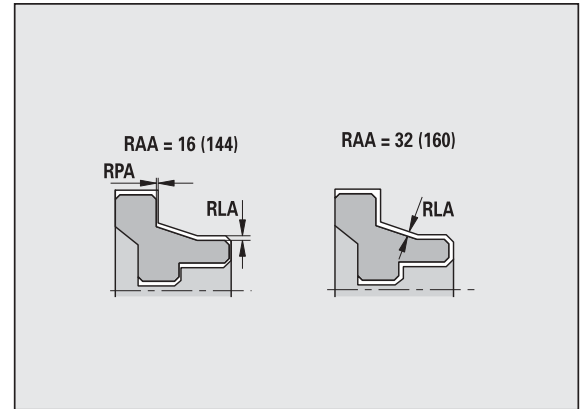
For tool selection, the following applies:

- Setting angle (EW): $EW \geq mkw$ (mkw: increasing contour angle)
- Setting angle (EW) and point angle (SW): $NWmin < (EW+SW) < NWmax$
- Secondary angle (RNWT): $RNWT = NWmax - NWmin$
- **Secondary angle tolerance [RNWT]** – tolerance range for secondary cutting edge
- **Relief cutting angle [RFW]** – Minimum angle differential between the contour and secondary cutting edge



The finished-part area can be assigned allowances:

- **Type of allowance [RAA]**
 - 16: Different longitudinal/transverse allowances – individual allowances are not considered
 - 144: Different longitudinal/transverse allowances – individual allowances are considered
 - 32: Equidistant allowance, individual allowances are not considered
 - 160: Equidistant allowance - individual allowances are considered
- **Equidistant or longitudinal [RLA]:** Equidistant allowance or longitudinal allowance
- **None or transverse [RPA]:** Transverse allowance



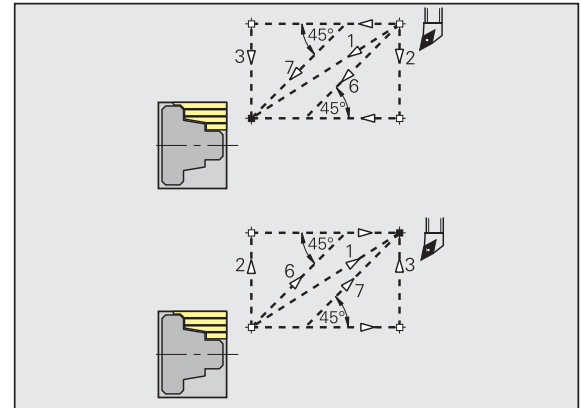
Roughing – Traverse from/to tool change position

- **Approach, external roughing [ANRA]**
- **Approach, internal roughing [ANRI]**
- **Depart, external roughing [ABRA]**
- **Depart, internal roughing [ABRI]**

Entries for retracting a tool:

- 1: Simultaneous, X and Z direction
- 2: First X direction, then Z
- 3: First Z direction, then X
- 6: Coupled motion, first X, then Z
- 7: Coupled motion, first Z, then X

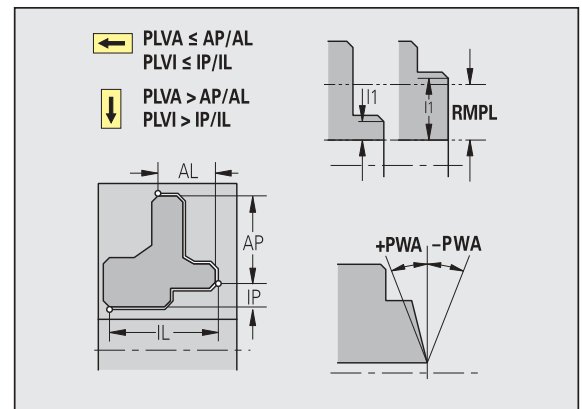
Approach and departure are in rapid traverse (G0).



Roughing – Machining analysis

TURN PLUS uses the PLVA/PLVI parameters to define whether a roughing area is to be rough-machined longitudinally or transversely.

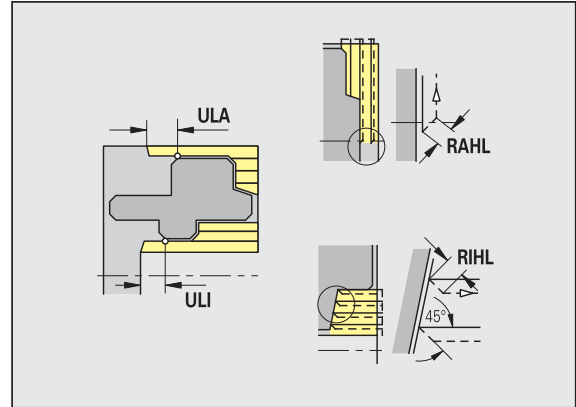
- **Transverse/longitudinal ratio - external [PLVA]**
 - $PLVA \leq AP/AL$: Longitudinal machining
 - $PLVA > AP/AL$: Transverse machining
- **Transverse/longitudinal ratio, internal [PLVI]**
 - $PLVI \leq IP/IL$: Longitudinal machining
 - $PLVI > IP/IL$: Transverse machining
- **Minimum transverse length [RMPL]** (radius value): defines whether the front transverse element of a finished part will have the external contour roughed transversely.
 - $RMPL > l1$: Without additional transverse roughing
 - $RMPL < l1$: With additional transverse roughing
 - $RMPL = 0$: Special case
- **Transverse angle variation [PWA]:** The first front element is declared a transverse element when it is within +PWA and -PWA.



Roughing – Machining cycles

- **Overhang length - external [ULA]**: Relative length for external rough-machining enabling roughing beyond the target position in longitudinal direction. Not considered when the cutting limitation is in front of or within the overhang.
- **Overhang length inside [ULI]** (see also "6.15.5 Inside Contours")
 - Length of roughing beyond the target point during inside machining in longitudinal direction. Not considered when the cutting limitation is in front of or within the overhang.
 - ULI is used to calculate the drilling depth for centric predrilling.
- **Retracting length - external [RAHL]**
- **Retracting length - internal [RIHL]**
 - Retracting length for smoothing variants (H=1, 2) of roughing cycles G810 and G820 for external machining (RAHL) / internal machining (RIHL).
- **Cutting-depth reduction factor [SRF]** – For rough-machining with tools machining opposite to the main machining direction, the infeed value (cutting depth) is reduced. Calculation of infeed (P) for roughing cycles (G810, G820):

$$P = ZT * SRF$$
 (ZT: Infeed from the technology database)

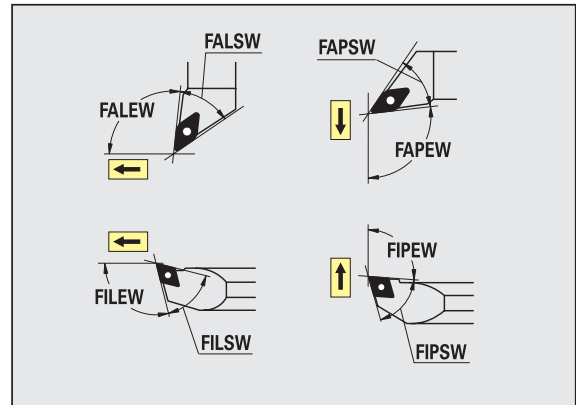


5 – Finishing

Finishing – Tool and Machining Standards

Finishing tools are defined according to machining location and main machining direction (MMD) via setting angle and point angle. For tool selection, the following additionally applies:

- Finishing cycles are primarily executed with standard finishing tools.
- If form elements such as recesses (type FD) and undercuts (type E, F, G) cannot be machined with a standard finishing tool, one form element after the other is skipped. TURN PLUS starts a renewed attempt to machine the remaining contour with the standard tool. Subsequently, the skipped form elements are machined individually with a suitable finishing tool.
- **Setting angle – external/longitudinal [FALEW]**
- **Point angle – external/longitudinal [FALSW]**
- **Setting angle – external/transverse [FAPEW]**
- **Point angle - external/transverse [FAPSW]**
- **Setting angle – internal/longitudinal [FILEW]**
- **Point angle - internal/longitudinal [FILSW]**
- **Setting angle – internal/transverse [FIPEW]**
- **Point angle - internal/transverse [FIPSW]**



The machining of contour areas is determined by the following parameters:

- **Standard/Complete – external/longitudinal [FAL]**
- **Standard/Complete – internal/longitudinal [FIL]**
- **Standard/Complete – external/transverse [FAP]**
- **Standard/Complete – internal/transverse [FIP]**

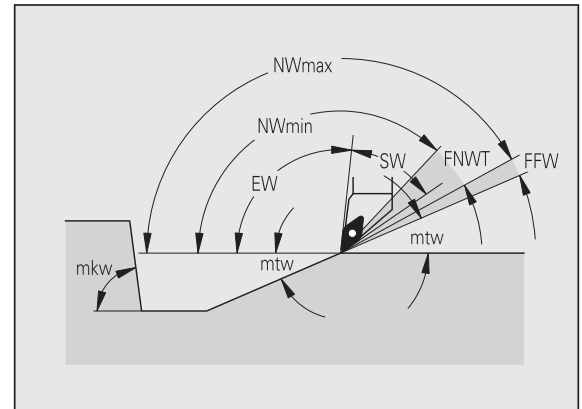
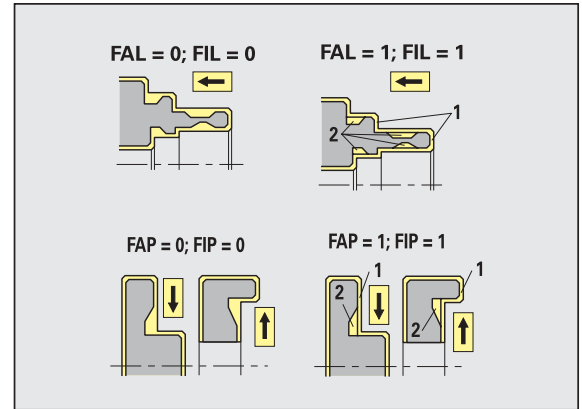
Entry:

- **0 – Complete finishing cycle:** TURN PLUS searches for an optimum tool for machining the complete contour area.
- **1 – Standard finishing cycle:**
 - is primarily executed with standard finishing tools. Relief turns and undercuts are machined with a suitable tool.
 - If the standard finishing tool is not suitable for machining relief turns and undercuts, TURN PLUS divides the machining process into standard machining cycles and the machining cycles for form elements.
 - If the division into standard machining cycles and machining cycles for form elements is not successful, TURN PLUS switches to "full-surface machining."

Finishing – Tool tolerances, traversing to/from tool change position

For tool selection, the following applies:

- Setting angle (EW): $EW \geq mkw$ (mkw: increasing contour angle)
- Setting angle (EW) and point angle (SW):
 $NWmin < (EW+SW) < NWmax$
- Secondary angle (FNWT): $FNWT = NWmax - NWmin$
- **Secondary angle tolerance [FNWT]** – tolerance range for secondary cutting edge
- **Relief cutting angle [FFW]** – Minimum angle differential between the contour and secondary cutting edge

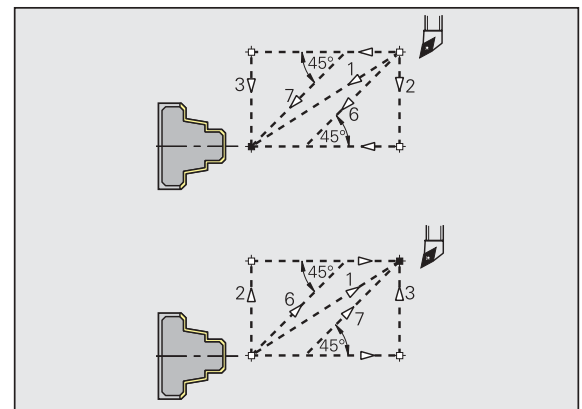


- **Approach - external finishing [ANFA]**
- **Approach - internal finishing [ANFI]**
- **Depart - external finishing [ABFA]**
- **Depart - internal finishing [ABFI]**

Entries for retracting a tool:

- 1: Simultaneous, X and Z direction
- 2: First X direction, then Z
- 3: First Z direction, then X
- 6: Coupled motion, first X, then Z
- 7: Coupled motion, first Z, then X

Approach and departure are in rapid traverse (G0).



Finishing – Machining analysis

■ **Minimum finishing transverse length [FMPL]** – TURN PLUS checks the frontmost element of the outside contour to be finish-machined. The following rules apply:

- Finished part with inside contour:
 - $FMPL \geq l1$: Without additional transverse cut
 - $FMPL < l1$: With additional transverse cut
- Finished part without inside contour: Always with additional transverse cut

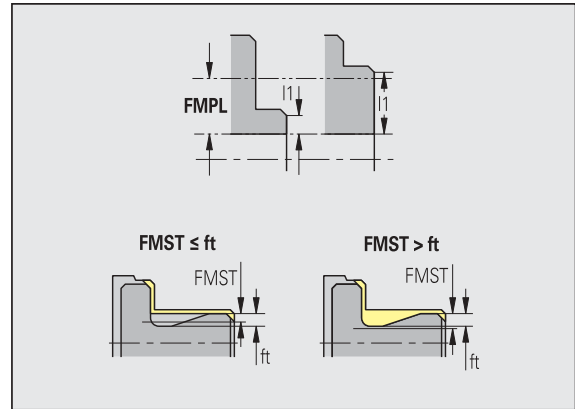


- The additional transverse cut is always from the outside to the inside.
- The "transverse angle variation PWA" does not influence the analysis of the transverse elements.

■ **Maximum finishing cut depth [FMST]** – defines the permissible infeed depth for non-machined undercuts. The finishing cycle (G890) uses this parameter to determine whether undercuts (type E, F, G) will be machined with a contour-finishing operation. The following applies:

- $FMST > ft$: With undercut machining (ft : undercut depth)
- $FMST \leq ft$: Without undercut machining

■ **Number of revolutions for chamfer or rounding [FMUR]** – The feed rate is reduced such that at least FMUR revolutions can be executed (evaluation: finishing cycle G890).



6 – Recessing and contour recessing

Recessing – Traverse from/to tool change position

- Approach - external recessing [ANESA]
- Approach - internal recessing [ANESI]
- Depart - external recessing [ABESA]
- Depart - internal recessing [ABESI]

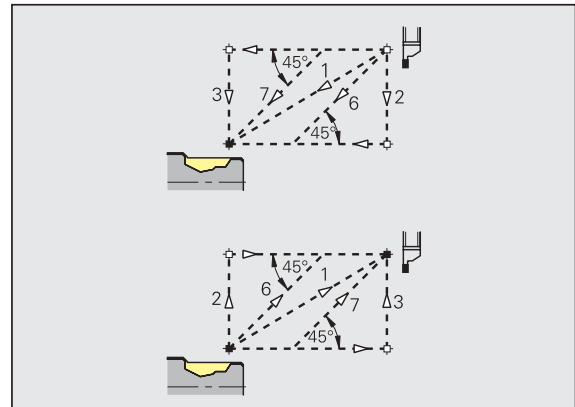
Recessing – Traverse from/to tool change position

- Approach - external contour recessing [ANKSA]
- Approach - internal contour recessing [ANKSI]
- Depart - external contour recessing [ABKSA]
- Depart - internal contour recessing [ABKSI]

Entries for retracting a tool:

- 1: Simultaneous, X and Z direction
- 2: First X direction, then Z
- 3: First Z direction, then X
- 6: Coupled motion, first X, then Z
- 7: Coupled motion, first Z, then X

Approach and departure are in rapid traverse (G0).



Recessing, contour recessing – Tool selection, allowances

If a recess base machined with a **contour-recessing** cycle does not contain paraxial elements but only linear elements, a suitable tool is selected using the "recessing width divisor SBD."

■ Recessing width divisor [SBD]

$SB \leq b / SBD$ (SB: width of recessing tool; b: width of machining area)

■ **Type of allowance [KSAA]**– The recessing area to be machined can be assigned allowances. When allowances have been defined, a recess is first rough-machined and then finish-machined. Entry:

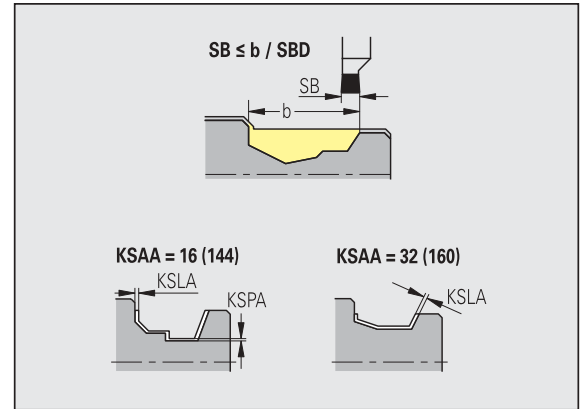
- 16: Different longitudinal/transverse allowances - individual allowances are not considered.
- 144: Different longitudinal/transverse allowances - individual allowances are considered.
- 32: Equidistant allowance – individual allowances are not considered.
- 160: Equidistant allowance - individual allowances are considered.

■ **Equidistant or longitudinal [KSLA]:** Equidistant allowance or longitudinal allowance

■ **None or transverse [KSPA]:** Transverse allowance



- The allowances are accounted for when machining contour valleys with a **contour-recessing** operation.
- Standardized recesses such as recess types D, S, A are completed in one machining cycle. A division into rough-machining and finish-machining is only possible in DIN PLUS.

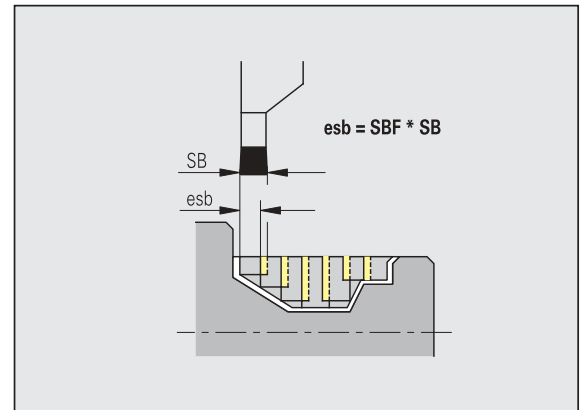


Recessing, contour recessing – Machining

Evaluation: DIN PLUS

■ **Recessing width factor [SBF]** – for determining the maximum offset during recessing cycles (G860, G866).

The following applies: $esb = SBF * SB$ (esb: effective recessing width (offset); SB: width of recessing tool)



7 – Thread cutting

Thread cutting - Traverse from/to tool change position

■ **Approach - external thread [ANGA]**

■ **Approach - internal thread [ANGI]**

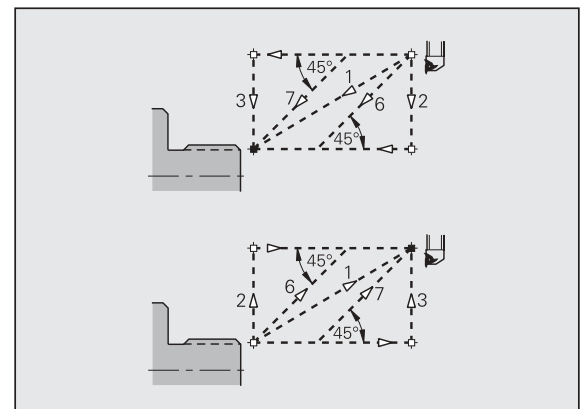
■ **Depart - external thread [ABGA]**

■ **Depart - internal thread [ABGI]**

Entries for retracting a tool:

- 1: Simultaneous, X and Z direction
- 2: First X direction, then Z
- 3: First Z direction, then X
- 6: Coupled motion, first X, then Z
- 7: Coupled motion, first Z, then X

Approach and departure are in rapid traverse (G0).

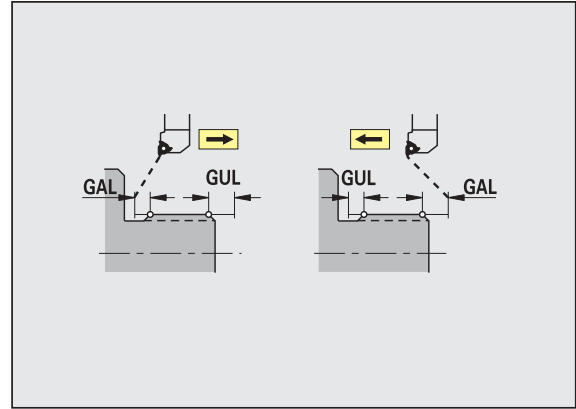


Thread cutting - Machining

- **Thread starting length [GAL]** – Starting length before threading cut
- **Thread run-out length [GUL]** – overrun length after threading cut



GAL/GUL are automatically transferred to the thread attributes "starting length B/overrun length P" if they have not been entered as attributes.



8 – Measuring

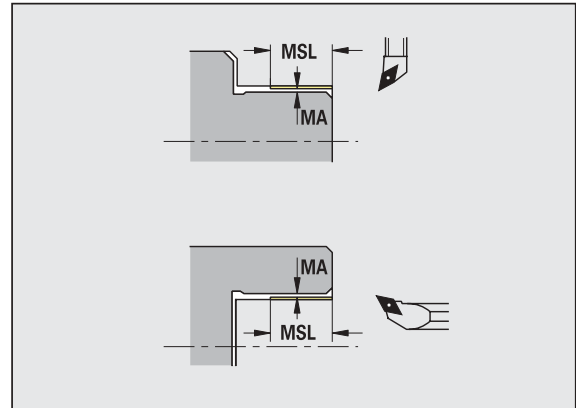
Measuring - Method of measurement

- **Measuring mode [MART]** – included.
 - 1: Manual measuring – calls an expert program
 - 2, 3: Currently not used
- **Measuring loop counter [MC]** – Defines the measurement/loop intervals.

Measuring – Measuring loop geometry

- **Measuring allowance [MA]** – Allowance still applied to the element to be measured.
- **Measuring cut length [MSL]**

The measuring parameters are assigned to the fit elements as an attribute.



9 – Drilling

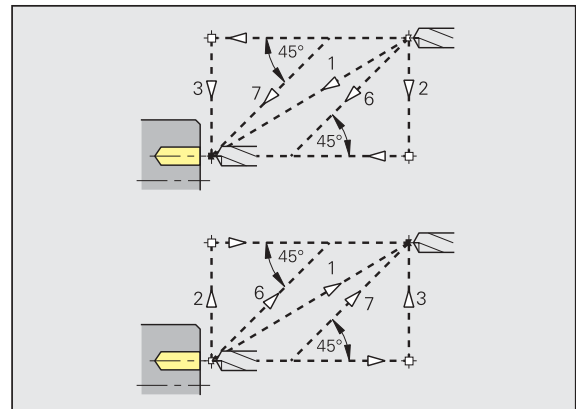
Drilling - Traverse from/to tool change position

- **Approach, end face [ANBS]**
- **Approach, lateral surface [ANBM]**
- **Depart, end face [ABGA]**
- **Depart, lateral surface [ABGI]**

Entries for retracting a tool:

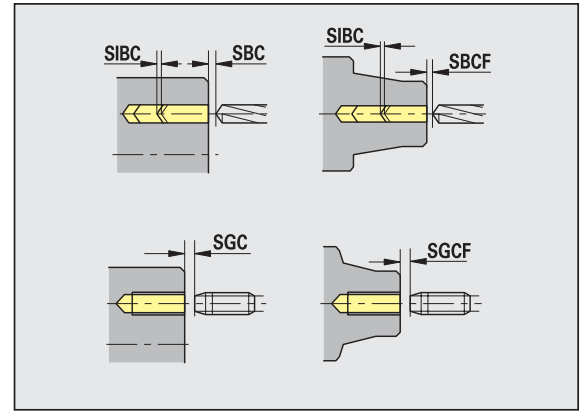
- 1: Simultaneous, X and Z direction
- 2: First X direction, then Z
- 3: First Z direction, then X
- 6: Coupled motion, first X, then Z
- 7: Coupled motion, first Z, then X

Approach and departure are in rapid traverse (G0).



Drilling - safety clearances

- **Internal safety clearance [SIBC]** – for deep-hole drilling (retraction distance B for G74).
- **Driven drilling tools [SBC]** – Safety clearance for driven tools on end face and lateral surface
- **Non-driven drilling tools [SBCF]** – Safety clearance for non-driven tools on end face and lateral surface
- **Driven tap [SGC]** – Safety clearance for driven taps on end face and lateral surface
- **Non-driven taps [SGCF]** – Safety clearance for non-driven taps on end face and lateral surface



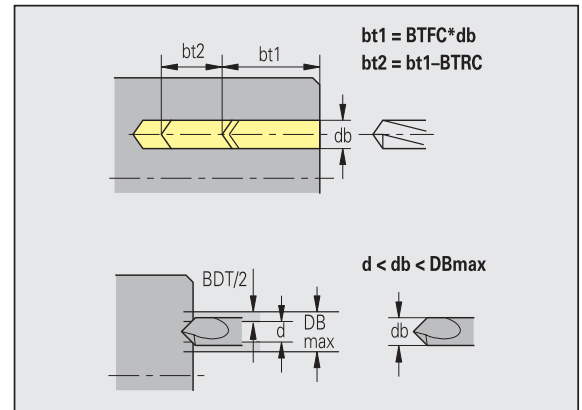
Drilling - Machining

The parameters apply to drilling with deep-hole drilling cycle (G74).

- **Drilling depth factor [BTFC]** – 1st drilling depth: $bt1 = BTFC * db$ (db: drill diameter)
- **Drilling depth reduction [BTRC]** – 2nd drilling depth: $bt2 = bt1 - BTRC$; the following drilling steps are reduced accordingly.
- **Diameter tolerance for drill [BDT]** – For selecting the desired drill (centering drills, countersinks, stepped drill, taper reamers).

Drilling diameter: $DBmax = BDT + d$ (DBmax: maximum drilling diameter)

Tool selection: $DBmax > DB > d$



10 – Milling

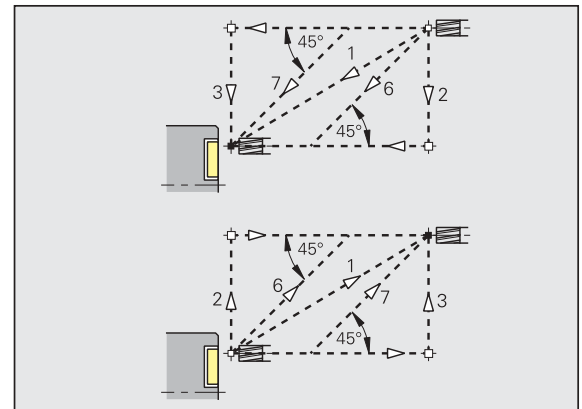
Milling - Traverse from/to tool change position

- **Approach, end face [ANMS]**
- **Approach, lateral surface [ANMM]**
- **Depart, end face [ABMA]**
- **Depart, lateral surface [ABMM]**

Entries for retracting a tool:

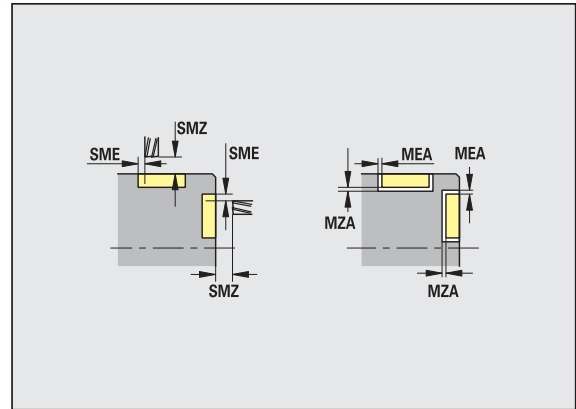
- 1: Simultaneous, X and Z direction
- 2: First X direction, then Z
- 3: First Z direction, then X
- 6: Coupled motion, first X, then Z
- 7: Coupled motion, first Z, then X

Approach and departure are in rapid traverse (G0).



Milling - Safety clearances and allowances

- **Safety clearance in infeed direction [SMZ]** – Distance between the starting position and the top edge of the workpiece to be milled.
- **Safety clearance in milling direction [SME]** – Distance between milling contour and cutter flank.
- **Allowance in milling direction [MEA]**
- **Allowance in infeed direction [MZA]**



11 – Load monitoring – General switches

- **Load monitoring ON/OFF**
 - 0: No load-monitoring commands generated by TURN PLUS.
 - 1: TURN PLUS generates commands for load monitoring.
- **Position, components**(corresponds to parameter Q in G996)
 - 0: Load monitoring not active.
 - 1: Rapid traverse paths are not monitored.
 - 2: Rapid traverse paths are monitored.

12..19 – Load monitoring for machining modes

The first parameter switches load monitoring on and off. The other parameters define the components to be monitored with regard to machining location/machining mode.

Entry for parameters 12..19:

- "Machining mode ..." on/off:
 - 0: Load monitoring off
 - 1: Load monitoring on
- Component to be monitored (for more than one component, the sum of the codes):
 - 0: No monitoring
 - 1: X axis
 - 2: Y axis
 - 4: Z axis
 - 8: Main spindle
 - 16: Driven tool
 - 32: Spindle 3
 - 64: Spindle 4
 - 128: C axis 1

12..19 – Load monitoring for

machining modes (continued)

- **12 Load monitoring, centric predrilling**
 - Drilling, centric ON/OFF
 - Centering
 - Drilling
 - Counterboring
 - Countersinking
 - Reaming
 - Tapping
- **13 Load monitoring, roughing**
 - Roughing ON/OFF
 - External, longitudinal
 - External, transverse
 - Internal, longitudinal
 - Internal, transverse
- **14 Load monitoring, contour recessing**
 - Piercing ON/OFF
 - External
 - Internal
 - Transverse
- **15 Load monitoring, contour machining**
 - Finishing ON/OFF
 - External
 - Internal
- **16 Load monitoring, recessing**
 - Recessing ON/OFF
 - External
 - Internal
- **17 Load monitoring, thread cutting**
 - Thread cutting ON/OFF
 - External
 - Internal
 - Transverse
- **18 Load monitoring, drilling C-axis**
 - Drilling, C-axis ON/OFF
 - Centering
 - Drilling
 - Counterboring
 - Countersinking
 - Reaming
 - Tapping
- **19 Load monitoring, milling C-axis**
 - Milling ON/OFF
 - Slot milling
 - Contour milling
 - Pocket milling
 - Deburring
 - Engraving

20 – Direction of rotation for rear-side machining

■ Mirror direction of rotation

- 0: Direction of spindle rotation remains the same for front and rear side
- 1: Direction of spindle rotation is mirrored, i.e. M3 becomes M4 and vice versa)

21 – Name of the subroutines

TURN PLUS uses expert programs for functions such as workpiece transfer for full-surface machining. This parameter defines which expert programs (subroutines) are used.

Enter the subprogram names.

- **UP 100098 (parting)**
- **UP 100099 (bar loader)**
- **UP EXUMS12 (currently no meaning)**
- **UP EXUMS12A (currently no meaning)**
- **UP MEAS01 (measuring cut)**
- **UP UMKOMPL (rechucking for lathes with counterspindle)**
- **UP UMKOMPLA (parting and rechucking for lathes with counterspindle)**
- **UP UMHAND (rechucking for lathes without counterspindle)**
- **UP ABHAND (parting and rechucking on machines without opposing spindle)**



8

Operating Resources

8.1 Tool Database

The CNC PILOT stores up to 999 tool descriptions which are managed via the tool editor.

Data exchange and data backup

The CNC PILOT supports **data exchange** and **data backup** of operating resources (tools, chucking equipment, technology data) and the associated fixed-word list—see “10 Transfer.”

8.1.1 Tool Editor

Select “Config” (Parameter mode of operation).

Editing the tool data

The tool data are edited in three dialog boxes. The parameters of the first two dialog boxes depend on the tool type. The third dialog box serves for multitool management and tool life management. Edit the third dialog box “as required.”

The tool parameters include:

- Basic data
- Information on tool depiction (simulation/control graphic)
- Information for TURN PLUS (tool selection, automatic working plan generation).

The tool data can be omitted provided that you do not use TURN PLUS, or do not wish tools to be depicted.

■ “New direct” menu item

- ▶ Enter the “tool type”.
- ▶ If the tool type is unknown, in the
 - Main group
 - Sub group
 - Machining directionpress the “continue” soft key and select the type and machining direction from the tool data
- ▶ Enter the tool data.

■ “New menu” menu

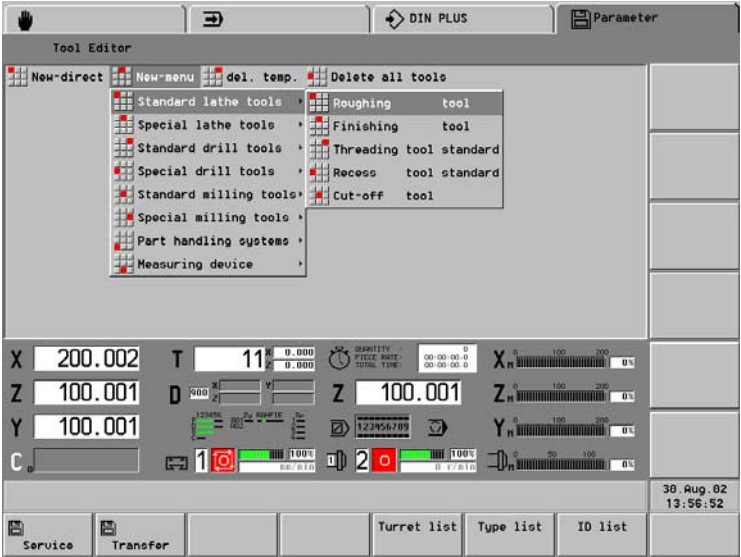
- ▶ Enter the tool type.
- ▶ Enter the tool data.

■ “del. temp.” menu item

Deletes tool descriptions that were recorded “temporarily” by NC program. Temporary tool descriptions start with “_SIM.” or “_AUTO.” (see “4.6.2 REVOLVER x”).



Tools that do not fit into any of the standard tool type groups are assigned to special turning/drilling/milling tools. They are not used for contour cycles and are not used by TURN PLUS.



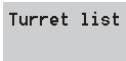
Soft keys



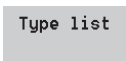
Changes to Service Mode



Switches to Transfer mode of operation



Registered tools of the tool carrier (turret)



Entries in the tool database—sorted by tool type



Entries in the tool database—sorted by tool ID number

Tool lists

Use the tool lists as a starting point for editing, copying or deleting entries.

Turret list

Lists the current tool carrier assignments.

Type list

Lists the entries sorted by tool type.
► Enter the "Tool type."
► Tool type unknown:
■ Main group
■ Subgroup
■ Machining direction
Select with "continue" soft key

ID list

Lists the entries, sorted by ID number.
"Mask for ID numbers" limits the list.
Only those entries are displayed that correspond to the mask.

Mask:
■ Enter the first characters of the ID:
The following places can be any characters.
■ ?: Any character can be at these positions.

Abbreviations (in header of the tool list):

- **rs**: Cutter radius
- **db**: Drill diameter
- **df**: Milling cutter diameter
- **ew**: Setting angle
- **bw**: Drilling angle
- **fw**: Milling angle
- **T no.** (T number in the turret list): See "4.2.4 Tool Programming"

Editing the tool list

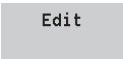
► Place the cursor on the desired tool and press the soft key.



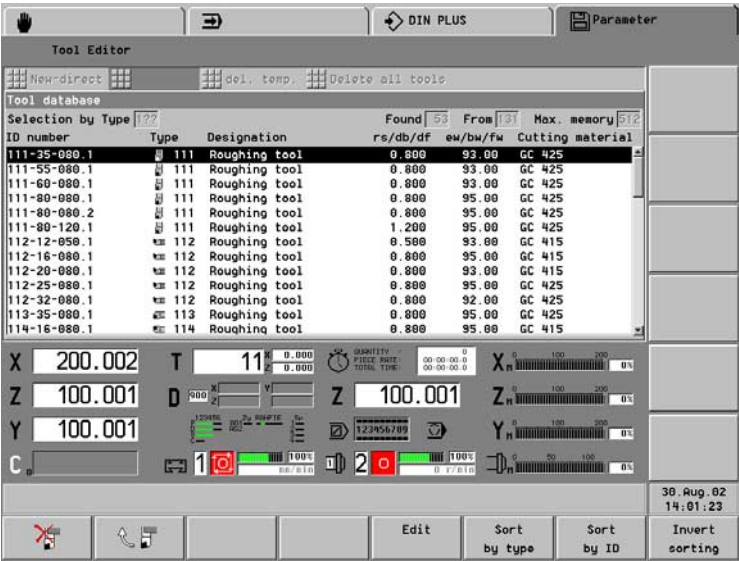
Copy entry
■ You can only copy "similar" tools
■ The new tool receives a new ID number



Delete entry



Or ENTER: Edit the entry



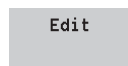
Soft keys



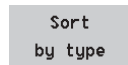
Delete tool entry



Copy tool entry



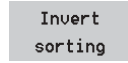
Edit tool entry



Sort displayed entries according to tool type



Sort displayed entries according to tool ID number



Reverse the sorting sequence



Display the tool graphic




Entries of the turret list are neither copied nor deleted in the tool editor. It is possible to edit the entries when Automatic mode is not active.

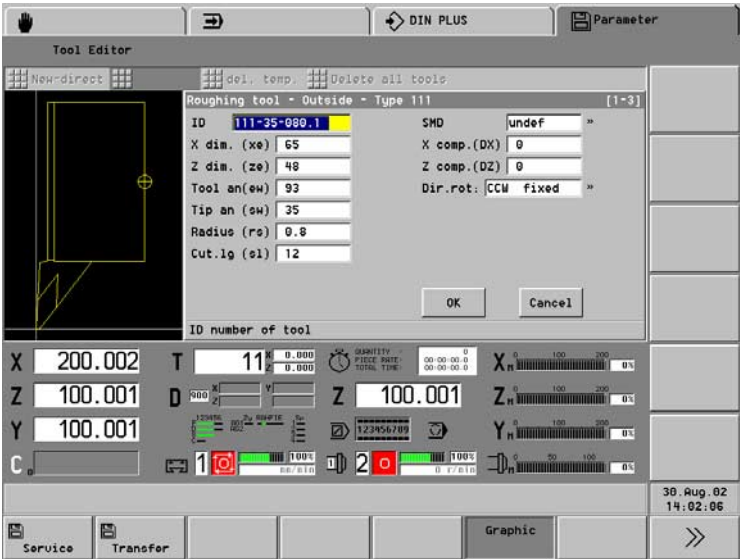
Continued ►

Display picture of tool

Graphic The CNC PILOT generates the displayed tool from the parameters. The graphic enables you to check the entered data. Changes become effective as soon you exit the input box.

To exit the graphic, press the soft key again

 **Tool position:** If the "mount type" tool parameter is used: The CNC PILOT looks for the mount type in the tool mount descriptions as of machine parameter 511. The first tool holder with this mount type is definitive for the tool position.

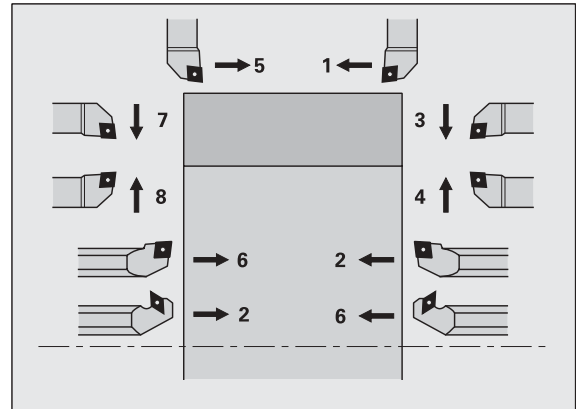


8.1.2 Tool Types (Overview)

Main machining direction (third digit of tool type): see figure.

Turning tools

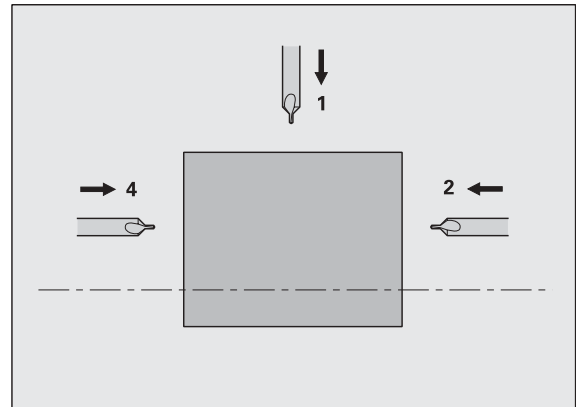
- Roughing tool (type 11x)
- Finishing tool (type 12x)
- Threading tool, standard (type 14x)
- Recessing tool (type 15x)
- Parting tool (type 161)
- Button tool (type 21x)
- Copying tool (type 22x)—TURN PLUS uses copying tools only for undercut types H and K.
- Recess turning tool (type 26x)
- Knurling tool (type 27x)
- Special turning tool (type 28x)



Example: Tool type 11x

Drilling tools

- Centering tool (type 31x)
- NC center drill (Type 32x)
- Twist drill (type 33x)
- Indexable-insert drills (type 34x)
- Counterbore (type 35x)
- Countersink (type 36x)
- Tap (type 37x)
- Step drill (type 42x)
- Reamer (type 43x)
- Tap drill (type 44x)
- Delta drill (type 47x)
- Turn-out tool (Type 48x)—not used by TURN PLUS
- Special drilling tool (type 49x)



Example: Tool type 31x

Milling Tools

- Twist drill cutter (type 51x)
- End milling cutter (type 52x)
- Side milling cutter (type 56x) – not used by TURN PLUS
- Angle cutter (type 61x)
- Thread milling cutter (type 63x) – not used by TURN PLUS
- Milling pins (type 64x)
- Circular saw blade (type 66x) – not used by TURN PLUS
- Special milling tool (type 67x)



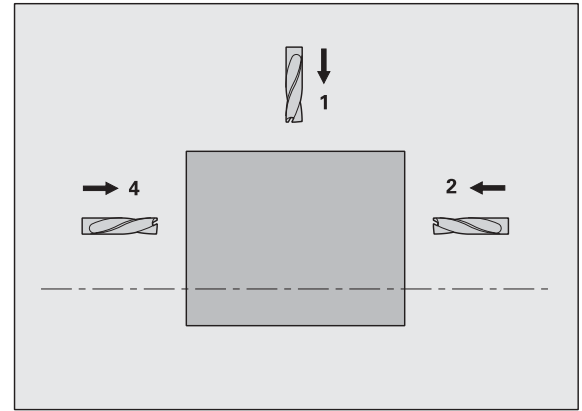
Special tools are tools that do not fit into any other type. They are not used for contour cycles and are not used by TURN PLUS..

Part Handling Systems

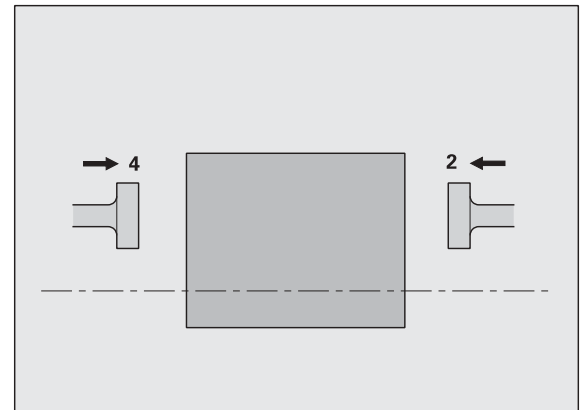
- Stopper tool (type 71x)
- Bar gripper (type 72x)
- Rotating gripping device (type 75x)

Measuring Devices

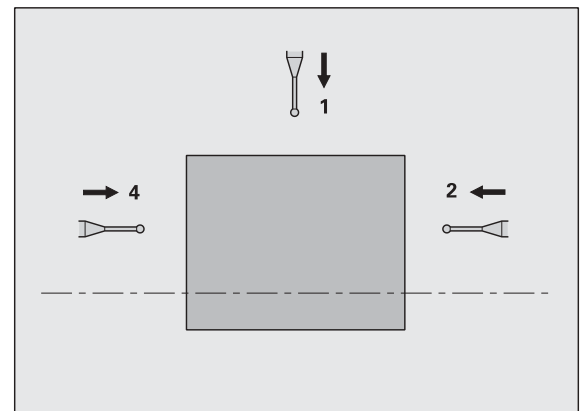
- Touch probe (type 81x)



Example: Tool type 51x



Example: Tool type 71x



Example: Tool type 81x

8.1.3 Tool Parameters

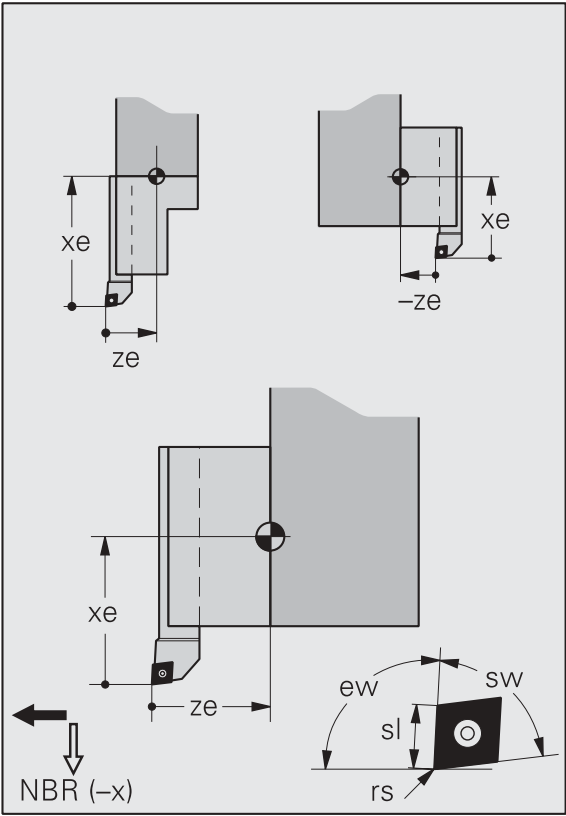
Parameters for turning tools

Parameters in dialog box 1	G	S	TP
ID: Identification number of tool	•	•	•
X, Z dim. (xe, ze): Setting dimensions	•	–	–
Tool angleW (ew): Setting angle	•	•	•
Tip angleW (sw): Nose angle	•	•	•
Radius (rs): Cutting radius	•	•	•
NBR: Secondary machining direction	•	–	•
Cut.wd (sb) – Thread tool: Cutting edge width – distance from the tool edge to the tool tip	•	•	–
Cut.wid. (sb): Cutting width	•	•	•
Cut.len. (sl): Cutting length	•	•	•
Cut.lg (sl) – Knurling tool: Roll diameter	–	•	–
RCut.wd (sb) – Knurling tool: Roll diameter	–	•	–
NBR: Secondary machining direction	•	–	•
X, Z comp. (DX, DZ): Compensation values (maximum value +/- 10 mm)	•	–	–
Direc.rot.: Direction of spindle rotation	•	–	•
Usab.lg. (nl): Usable length for internal too ls –	–	•	
Dep.imm. (et): Maximum infeed depth	•	•	•
3 comp. (DS): Special correction value for the 3rd cutting edge (maximum value +/- 10 mm) – see also G148 and G150/G151	•	–	–



Threading tool:

- Note that for types 141, 143 the "setting dimension ze" and for type 142, 144, the "setting dimension xe" is measured from the tool edge.
- The CNC PILOT uses the "direction of rotation" parameter to determine whether an overhead tool or a standard tool is used.



Example: Tool type 111

Continued ►

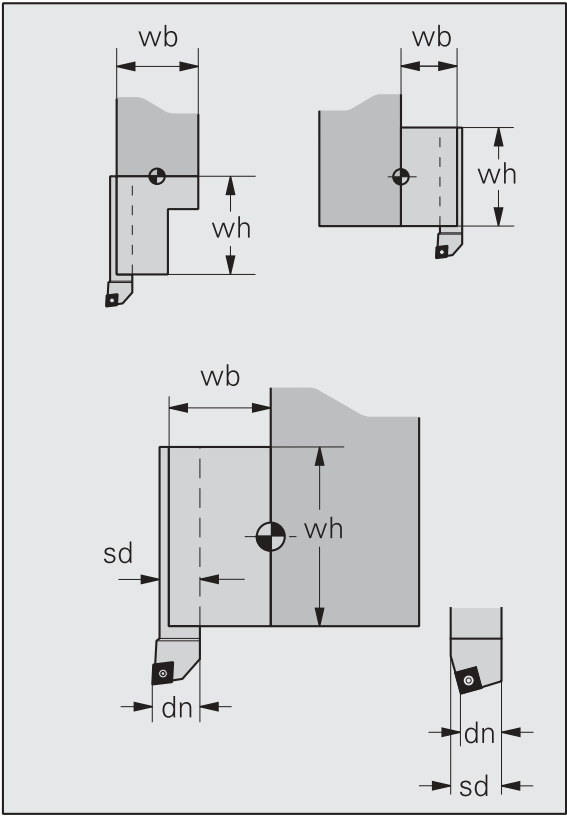
Parameters in dialog box 2	G	S	TP
TH DIN: Type of tool holder	–	•	–
TH heig (wh): Height of tool holder	–	•	–
TH brea. (wb): Width of tool holder	–	•	–
Breadth (dn): Tool width (tool tip to shank back)	–	•	–
Shank d (sd): Shank diameter	–	•	–
Design (A): Left-hand or right-hand tool	•	•	•
Design (A) – Button tools: Left, right or neutral tool versions in the tool positions 1..4	•	•	•
Pitch: Thread pitch	•	–	•
Available.: Physical availability	•	–	•
Pict. no. (tool display)	–	•	–
Cut.mat. Cutting material	–	–	•
CSP comp.: Compensation factor for cutting speed	–	–	•
FDR comp.: Compensation factor for feed rate	–	–	•
Deep comp.: Compensation factor for cutting depth	–	–	•
Location type	•	–	•

G: Basic data
S: Tool depiction (simulation)
TP: TURN PLUS

- See also:
- "8.1.4 MultipleTools, Tool Life Monitoring" (parameters of the third dialog box)
 - "8.1.5 Notes on Tool Data"
 - "8.1.6 Tool Holder, Mounting Position"



- The "version" parameter defines whether the tool reference point lies on the right or left side of the cutting edge.
- For neutral button tools, the tool reference point lies on the left side of the cutting edge.



Example: Tool type 111

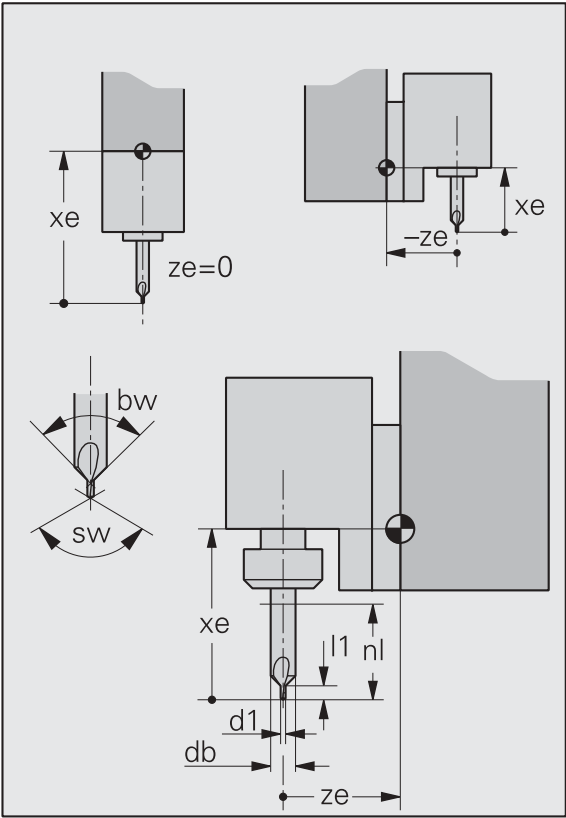
Drilling tool parameters

Parameters of dialog box 1	G	S	TP
ID: Identification number of tool	•	•	•
X, Z, Y dim. (xe, ze, ye): Setting dimensions	•	–	–
Diamet. (db): Drill diameter	•	•	•
Bor.ang. (bw): Drill angle	•	•	•
Point angleW (sw): Point angle	•	•	•
Shnk.di. (d1): Shank diameter	•	•	•
Shnk.ln. (l1): Shank length	•	•	•
Pos.ang. (rw): Position angle	•	•	–
X, Z, Y comp. (DX, DZ, DY): Compensation values (maximum value +/- 10 mm)	•	–	–
Direc.rot.: Direction of spindle rotation	•	–	•
Usab.lg.(nl): Usable length of drill	–	–	•
Type of tap: See fixed-word list	*1	–	*1
Strtlen (al): Length of first cut	•	•	•

Fixed-word list for "Tap type":

- 0: Undefined
- 11: Metric
- 12: Fine thread
- 13: Whitworth thread
- 14: Pipe thread
- 15: UNC
- 16: UNF
- 17: PG
- 18: NPT
- 19: Tetragonal thread
- 20: Other threads

*1: The "Type of tap" parameter is used for determining the thread parameters; the AWG accounts for this parameter when selecting a tool.



Example: Tool type 311

Continued ►

Parameters in dialog box 2	G	S	TP
TH DIN: Type of tool holder	–	•	–
TH heig (wh): Height of tool holder	–	•	–
TH brea. (wb): Width of tool holder	–	•	–
Chck.. (fd): Diameter of chuck	–	*1	–
Chck.he. (fh): Height of chuck	–	*1	–
Sali.lg.(ax): Overhang length	–	•	–
Pitch (hb): Thread pitch	•	–	•
Fit qual(ity): See fixed-word list *2	–	–	•
Available.: Physical availability	•	–	•
Pict. no. (tool display)	–	•	–
Cut.mat. Cutting material	–	–	•
CSP comp.: Correction factor for cutting speed	–	–	•
FDR comp.: Compensation factor for feed rate	–	–	•
Deep comp.: Compensation factor for cutting depth	–	–	•
Location type	•	–	•

Fixed-word list for fit quality:

- H6
- H7
- H8
- H9
- H10
- H11
- H12
- H13

*1 – Chuck dimensions

- Holder types F, K: "fd, fh" serve to indicate the holder dimensions
- Other holders: Where fd=0 and fh=0, the chuck is not displayed.

*2: TURN PLUS's automatic tool selection checks whether the "fit quality" is defined. There is no detailed evaluation.

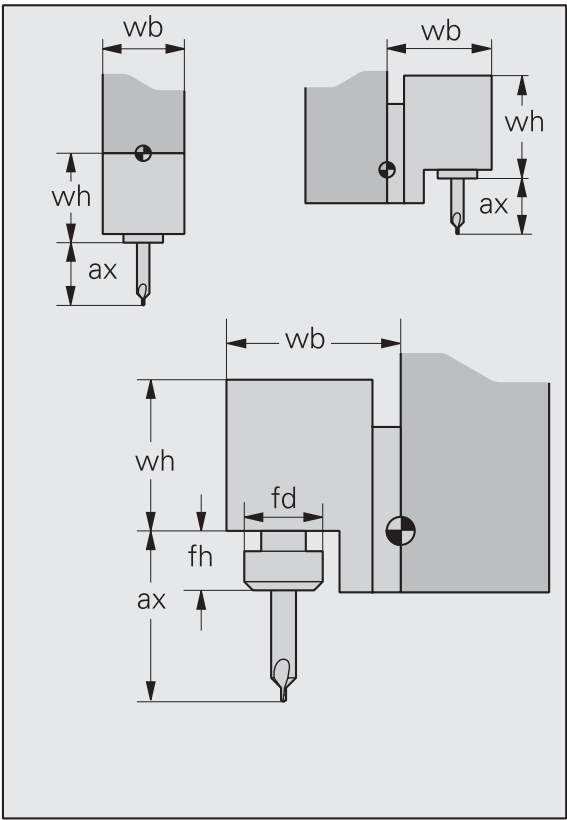
G: Basic data

S: Tool depiction (simulation)

TP: TURN PLUS

See also:

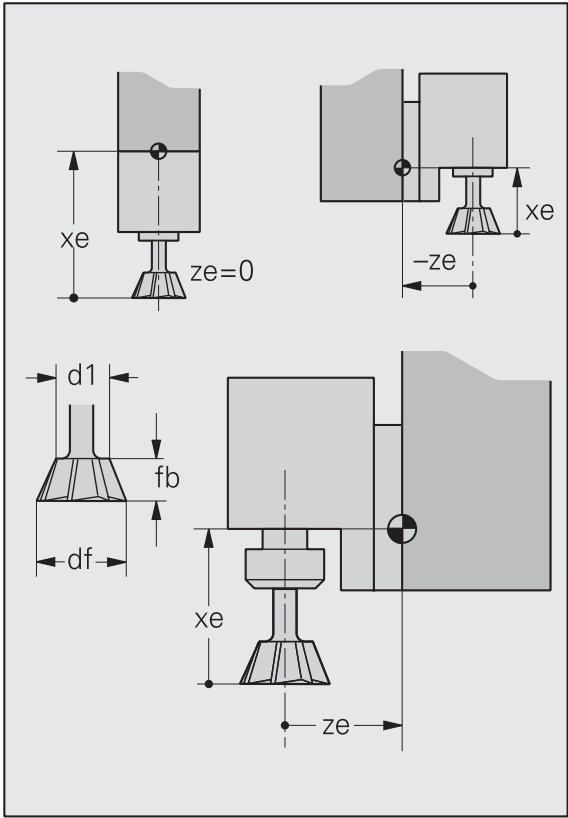
- "8.1.4 MultipleTools, Tool Life Monitoring" (parameters of the third dialog box)
- "8.1.5 Notes on Tool Data"
- "8.1.6 Tool Holder, Mounting Position"



Example: Tool type 311

Milling cutter parameters

Parameters in dialog box 1	G	S	TP
ID: Identification number of tool	•	•	•
X, Z, Y dim. (xe, ze, ye): Setting dimensions	•	–	–
Diamet. (df): Cutter diameter, end	•	•	•
Diamet. (d1): Cutter diameter	•	•	•
Width (fb): Cutter width	•	•	•
Angle (fw): Cutter angle	•	•	•
Dep.imm. (et): Maximum infeed depth	•	•	–
Pos.ang. (rw): Position angle	•	•	–
X, Z, Y comp. (DX, DZ, DY): Compensation values (maximum value +/- 10 mm)	•	–	–
D corr. (DD): Compensation value for cutter diameter	•	–	–
Direc.rot.: Direction of spindle rotation	•	–	•
Cut.len.(sl): Cutting length	•	•	•
Number of teeth of milling cutter	•	–	•



Example: Tool type 611

Continued ►

Parameters in dialog box 2	G	S	TP
TH DIN: Type of tool holder	–	•	–
TH heig (wh): Height of tool holder	–	•	–
TH brea. (wb): Width of tool holder	–	•	–
Chck.. (fd): Diameter of chuck	–	*1	–
Chck.he. (fh): Height of chuck	–	*1	–
Sali.lg.(ax): Overhang length	–	•	–
Pitch (hf): Thread pitch	•	–	–
Threads per unit length (gb) for multiple threads	–	–	–
Tooth pattern of cutter – see fixed word list	–	–	•
Available.: Physical availability	•	–	•
Pict. no. (tool display)	–	•	–
Cut.mat. Cutting material	–	–	•
CSP comp.: Compensation factor for cutting speed	–	–	•
FDR comp.: Compensation factor for feed rate	–	–	•
Deep comp.: Compensation factor for cutting depth	–	–	•
Location type	•	–	•

Fixed-word list for tooth pattern:

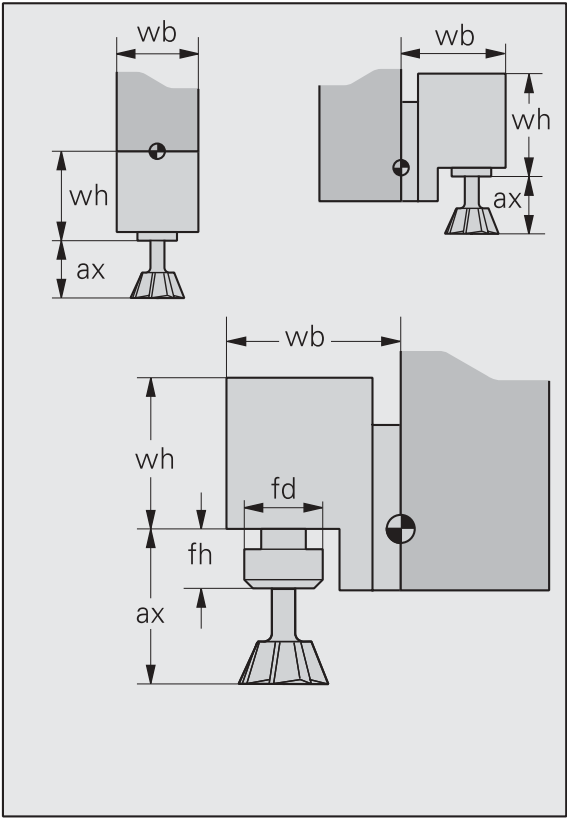
- 0: Undefined
- 1: SpurFac (straight, end)
- 2: HeliFac (helical, end)
- 3: SpurCir (straight, circumference)
- 4: HeliCir (helical, circumference)
- 5: SEndCir (straight, end and circumference)
- 6: HEndCir (helical, end and circumference)
- 7: Special tooth pattern

*1: Where fd=0/fh=0 no chuck is depicted.

- G: Basic data
S: Tool depiction (simulation)
TP: TURN PLUS

See also:

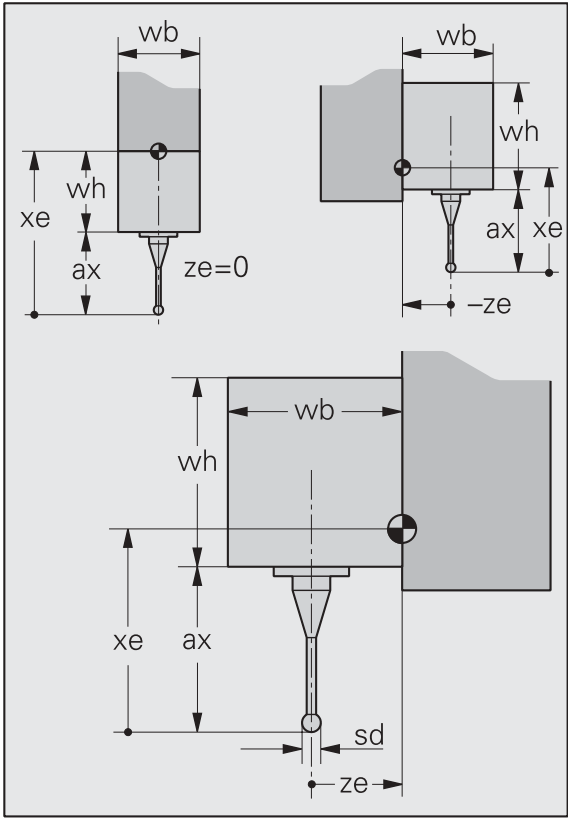
- "8.1.4 MultipleTools, Tool Life Monitoring" (parameters of the third dialog box)
- "8.1.5 Notes on Tool Data"
- "8.1.6 Tool Holder, Mounting Position"



Example: Tool type 611

Parameters for workpiece handling system and encoders

Parameters in dialog box 1	G	S	TP
ID: Identification number of tool	•	•	–
X, Z dim. (xe, ze): Setting dimensions	•	–	–
Available.: Physical availability	•	–	–
Shank d (sd): Shank diameter	–	•	–
Multiple tool: Multiple tool (see "4.2.4 Tool Programming")	•	–	–
■ No: No multiple tool			
■ Main: Primary cutting edge			
■ Aux.: Secondary cutting edge			
M ID: Identification number of the following cutter of a multiple tool	•	–	–
TH DIN: Type of tool holder	–	•	–
TH heig (wh): Height of tool holder	–	•	–
TH brea. (wb): Width of tool holder	–	•	–
Sali.lg.(ax): Overhang length	–	•	–
Pict. no. (tool display)	–	•	–
Location type	•	–	–
Mag(azine) code: Not used at present			
Mag(azine) attr(ibute): Not used at present			



Example: Tool type 811

8.1.4 Multipoint Tools, Tool Life Monitoring

Multipoint tools

Turning tools with more than one (maximal 5) cutting edges are referred to as multipoint tools. In the tool database, each cutting edge is given a tool definition—in addition, the identification numbers of the cutting edges of a Multipoint tool are linked so that a sequence is defined which comprises all cutting edges.

Define one of the cutting edges as **primary cutting edge** and the other ones as **secondary cutting edges**. In the tool list, the ID number of the primary cutting edge (see “4.2.4 Tool Programming”).

Parameters of 3rd dialog box

Mag(azine) code: Not used at present

Mag(azine) attr(ibute): Not used at present

Multi tool: Multipoint tool
■ **No:** No Multipoint tool
■ **Main:** Primary cutting edge
■ **Aux.:** Secondary cutting edge

M ID: Identification number of the following cutter of a multipoint tool

Mon(itoring) method of the tool life monitoring function (see “4.2.4 Tool Programming”)
■ **No**
■ **Tool life monitoring**
■ **Quantity monitoring**

Tool life total: Tool life of the cutting edge

Tool life rem.: Display of remaining tool life

Quantity total: Total quantity that can be produced by a tool.

Quantity remaining: Display of remaining quantity.


Reason for retiring Displays reason tool has been retired:
■ Tool life expired
■ Quantity reached
■ Tool life expired
 ■ Determined by in-process measuring
 ■ Determined through postprocess measurement
■ Tool wear (limit value 1 or 2 of the power has been exceeded)—detected through load monitoring
■ Tool wear (limit value of “work” exceeded)—determined by load monitoring

The tool life parameters are reset with a new cutting edge (see “3.5.5 Tool Life Management”).

Data input for multipoint tools

- Primary cutting edge:
- ▶ Parameter input (dialog box 1 and 2)
 - ▶ Use the “PgUp” key to switch to dialog box 3
 - ▶ Enter the **Main (cutting edge)** in the “Multi. tool” input field
 - ▶ Enter the identification number of the **following** secondary cutting edge in the “M ID” input field
 - ▶ Conclude with “OK.”

- For each secondary cutting edge:
- ▶ Enter the identification number (ID number that has been entered into the “M ID” field for the preceding cutting edge)
 - ▶ For further parameter input (dialog boxes 1 and 2)
 - ▶ Use the “PgUp” key to switch to dialog box 3
 - ▶ Enter the **secondary (cutting edge)** in the “Multi. tool” input field
 - ▶ Enter the identification number of the following secondary cutting edge in the “M ID” input field; for the **last secondary cutting edge**, enter the identification number of the main cutting edge
 - ▶ Conclude with “OK.”

 Watch for the “closed chain” in Multipoint tools (main tooth—auxiliary teeth—main tooth).

8.1.5 Explanation of Tool Data

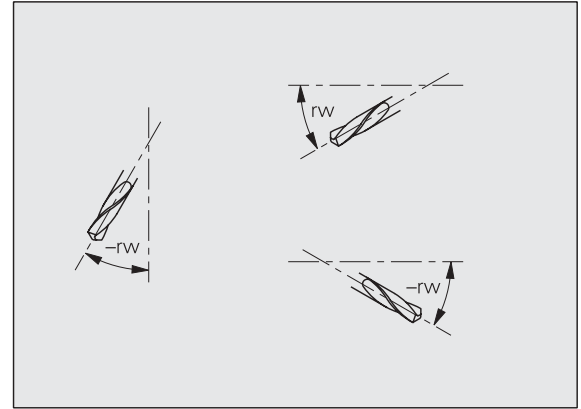
- **Tool ID number:** Each tool is assigned a unique ID (with up to 16 letters and numbers). The ID number must not start with an underscore.
- **Tool type:**
 - First and second numeral: Type of tool
 - Third numeral: Tool position/main machining direction
- **Setting dimensions (xe, ze):** Distance from tool reference point to tool-carrier reference point (Tool-carrier reference point: see machine manual)
- **Compensation values (DX, DZ, DS)** compensate for the wear of the cutting edge. For recessing and button tools, DS stands for the compensation value of the third side of the tool (away from the tool reference point).
- **Cutting length (sl):** Length of cutting insert.
 - Check the contour cycles as to whether the tool can perform the required cutting job.
 - Influences tool selection in TURN PLUS.
 - Is evaluated for simulating the cutting path and depicting the tool.
- **Secondary machining direction (NBR):** Defines the directions in which the tool works in addition to the main direction.
 - The contour cycles check whether the tool is suitable for the respective machining operation.
 - Influences the tool selection in TURN PLUS.
 - For the secondary machining direction, the AWG uses: The **secondary feed rate** (see "8.3 Technology Database (Cutting Values)")
 - a reduced cutting depth (see machining parameter 4 – "SRF")
- **Direction of rotation** defines the direction of spindle rotation for the tool; defines whether the tool is a driven/non-driven tool.
 - The contour-based cycles check whether the tool is suitable for the respective machining operation.
 - The direction of rotation influences tool selection in TURN PLUS.
 - It also defines the direction of spindle rotation for AWG.
- **Width (dn):** Tool width from tool tip to shank back. The parameter for width (dn) is evaluated for graphically representing the tool.
- (Physically) **available:** Identifies an unavailable tool without deleting the database entry.
- **Version:** "Left or right tool" – defines the position of the tool reference point. For neutral version tools, the tool reference point is located on the left side.
- **Picture number:** Display the tool or only the tool tip?
 - 0: Display tool
 - -1: Display only the tool tip



A ">>" after the input box means "fixed-word list." Select the tool parameters from the "fixed-word list" and use it as input.
 Calling the fixed-word list: Position the cursor on the input box and press the ">>" soft key.

Continued ►

- **CSP correction:** Cutting speed
- **FDR correction:** Feed rate
- **Deep correction:** Cutting depth
- TURN PLUS multiplies the cutting data from the technology database by the compensation values entered here.
- **Mount type** – is used for lathes with different tool holders. The tool is used if it has the same mount type as is defined for this pocket (see machine parameter 511, ...).
 - The "mount type" influences tool selection and positioning in TURN PLUS.
 - The function "Set up tool table" checks whether the tool can be used on the designated turret position.
- **Angle of orientation (rw):** Defines the deviation from the main machining direction in the mathematically positive direction of rotation ($-90^\circ < rw < +90^\circ$).
TURN PLUS only uses drilling and milling tools machining in direction of or at right angles to the principal axis.
- **Tooth number:** Used for "G93 feed rate per tooth"
- **Salient length (ax)** – for drilling and milling tools:
 - Axial tools: ax = Distance from tool reference point to the upper edge of the holder.
 - Radial tools: ax = Distance from tool reference point to the lower edge of the holder (also for drilling/milling tools clamped in a chuck).



Dimension of "Position angle rw"

8.1.6 Tool Holder, Mounting Position

The graphic tool representation functions (simulation and control graphics) take account of the shape of the tool holder and the mounting position on the tool carrier.

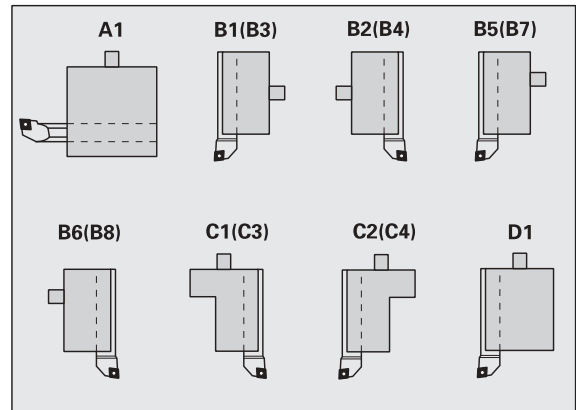
Tool holder

Depending on the turret location, the CNC PILOT determines whether the holder is mounted in an axial or radial position or whether an adapter is used.

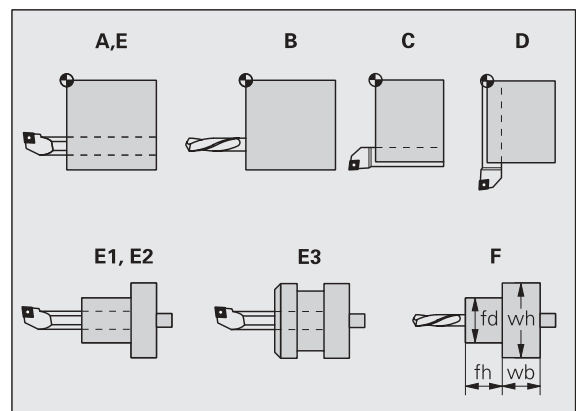
When the tool holder type is not specified, the CNC PILOT uses a simplified graphic representation.

The CNC PILOT recognizes the following holders (designation of the standard holder as per DIN 69 880):

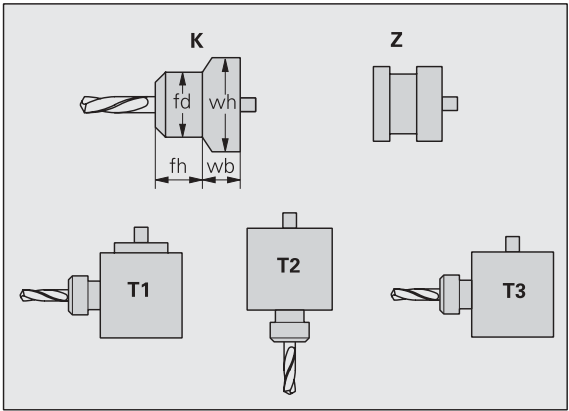
- A1 three-bar boring tool
- B1 right-hand, short design
- B2 left-hand, short design
- B3 right-hand, short design, overhead
- B4 left-hand, short design, overhead
- B5 right-hand, long design
- B6 left-hand, long design
- B7 right-hand, long design, overhead
- B8 left-hand, long design, overhead
- C1 right-hand
- C2 left-hand
- C3 right-hand, overhead
- C4 left-hand, overhead
- D1 multiple holder



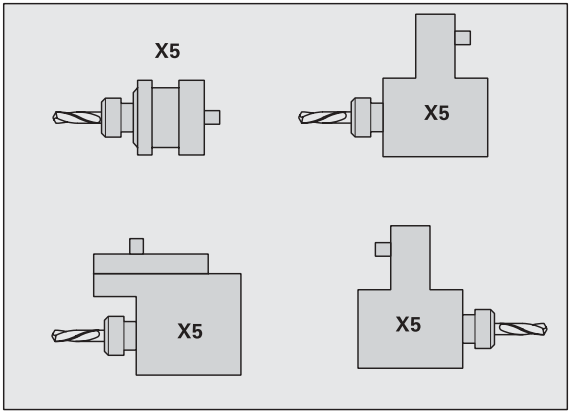
- A three-bar boring tool
- B drill holder with coolant circulation
- C square, longitudinal
- D square, transverse
- E end-face machining
- E1 U drill
- E2 cylindrical-shank holder
- E3 collet holder
- F drill holder MK (Morse taper)



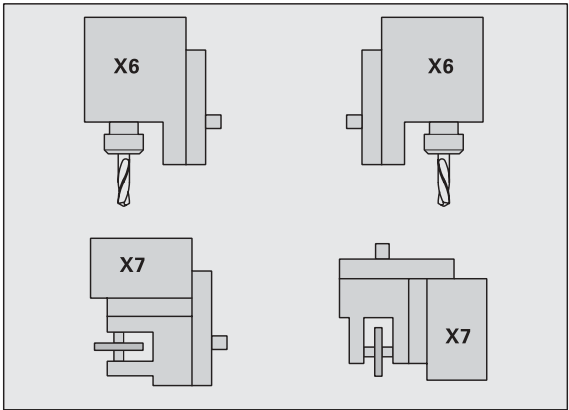
- K drill chuck
- Z stop
- T1 driven, axial
- T2 driven, radial
- T3 three-bar boring tool



- X5 driven, axial

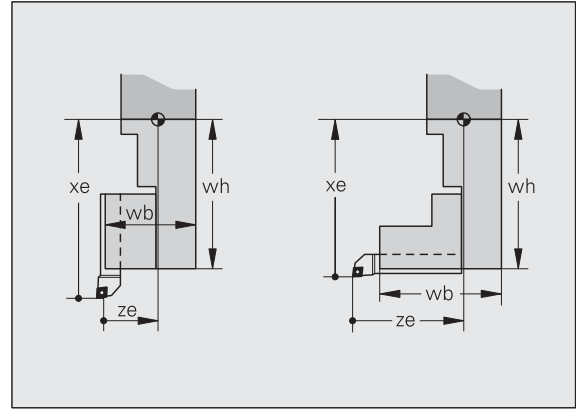


- X6 driven, radial
- X7 driven, special holder



Adapter

When an adapter is used, the values entered for tool height (wh) and tool width (wb) refer to the height/width of the adapter **and** the holder.



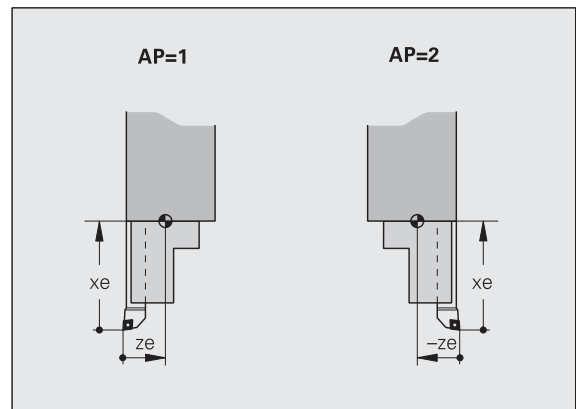
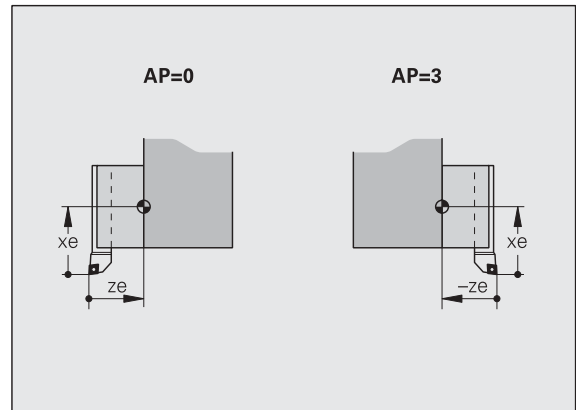
Tool mounting position

The mounting position is fixed by the machine tool builder (see Machine Parameter 511, ...). Depending on the turret location, the CNC PILOT determines the tool mounting position.

- Axial mount – left turret side (AP=0)
- Radial mount – left turret side (AP=1)
- Radial mount – right turret side (AP=2)
- Axial mount – right turret side (AP=3)



When the tool is mounted radially in the middle of the turret, "AP=1" is used.



8.2 Chucking Equipment Database

The CNC PILOT stores up to 999 chucking equipment descriptions which are managed via the chucking equipment editor.

Chucking equipment is used in TURN PLUS mode and displayed in the simulation/control graphics.

Chucking equipment identification number

Each chuck has its own ID number (up to 16 characters/letters). The ID number must not start with with an underline character.

Chucking equipment type

The chuck type defines the type of chuck/jaws.

8.2.1 Chucking Equipment Editor

Select “Chuck”(ing equipment) menu item (Parameters mode)

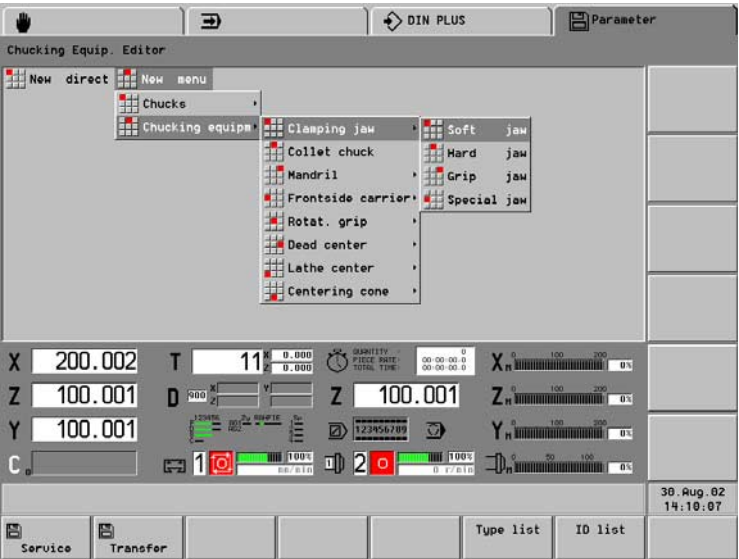
Editing chuck data

Chuck data are edited in a dialog box. Chuck parameters contain data for chuck representation in the simulation/control graphics, and further data for chuck selection in TURN PLUS.

The chuck parameters can be omitted provided that you do not use TURN PLUS, or do not wish chucks to be displayed in the simulation graphics.

Chuck definition

- “New direct” menu item
 - ▶ Enter “chucking equipment type”
 - ▶ Enter the chucking equipment data in the dialog box
- “New direct” menu item
 - ▶ Enter “chucking equipment type” in the submenu
 - ▶ Enter the chucking equipment data in the dialog box



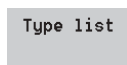
Soft keys



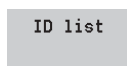
Changes to Service Mode



Switches to Transfer mode of operation



Entries in the chucking database—sorted by chuck type



Entries in the chucking database—sorted by chuck ID

Chucking equipment lists

The CNC PILOT lists the chucking equipment according to identification numbers or chuck types.

The chucking equipment list serves as starting point for editing, copying or deleting entries.

ID list

Lists the entries, sorted by ID number. "Mask for ID numbers" limits the list. Only those entries are displayed that correspond to the mask.

Mask:
■ Enter the first characters of the ID: The following places can be any characters.
■ ?: Any character can be at these positions.

Type list

Lists the entries sorted by chuck type. You can limit the list by using the "type number" mask. Only those entries are displayed that correspond to the mask.

The **chuck header** informs you of the mask entered, the number of chucks found and the number of saved chucks. In addition, the maximum number of chucks saved by the CNC PILOT is indicated.

Editing the chucking equipment list

Move the cursor to the desired chucking equipment and press the corresponding key.

Insert

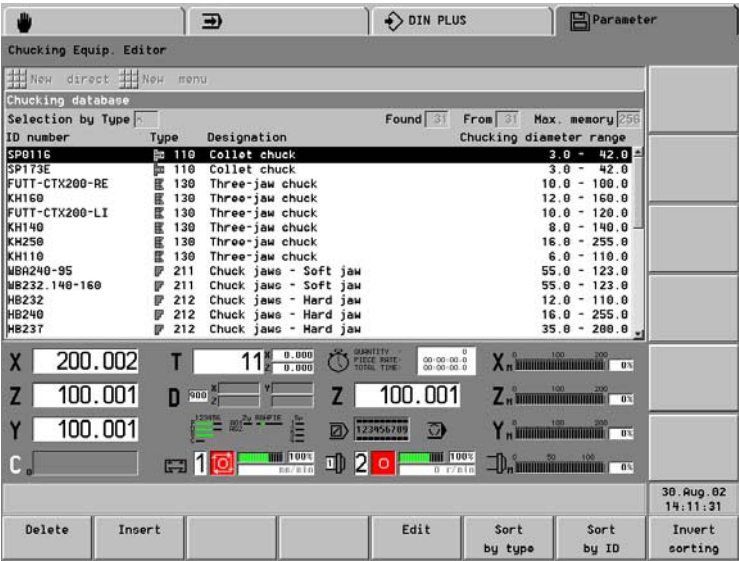
Copy the entry (only chucking equipment of the same type)

Delete

Delete entry

Edit

Or ENTER: Edit the entry



Soft keys

Delete

Delete a chuck entry

Insert

Copy chucking equipment entry

Edit

Editing a chuck entry

Sort by type

Sort displayed entries according to chuck type

Sort by ID

Sort displayed entries according to chuck ID number

Invert sorting

Reverse the sequence of sorting

8.2.2 Chucking Equipment Data

Overview of chuck types

The chuck parameters depend on the chuck types.

Primary chucking equipment

Chuck
Chucking equipment

Chuck	Type
Collet chuck	110
Two-jaw chuck	120
Three-jaw chuck	130
Four-jaw chuck	140
Face chuck	150
Special chuck	160

Chucking equipment	Type
Chuck jaws	21x
Soft jaws	211
Hard jaws	212
Grip jaw	213
Special jaw	214
Collet chuck	220
Mandril	23x
Face driver	24x
Rotating gripper	25x
Dead center	26x
Lathe center	27x
Centering cone	28x

Adapters for chucking types 23x..28x	Type
Cylindric chuck adapter	xx1
Plain flange adapter	xx2
Morse cone MK3	xx3
Morse cone MK4	xx4
Morse cone MK5	xx5
Morse cone MK6	xx6
Other adapters	xx7

Chuck

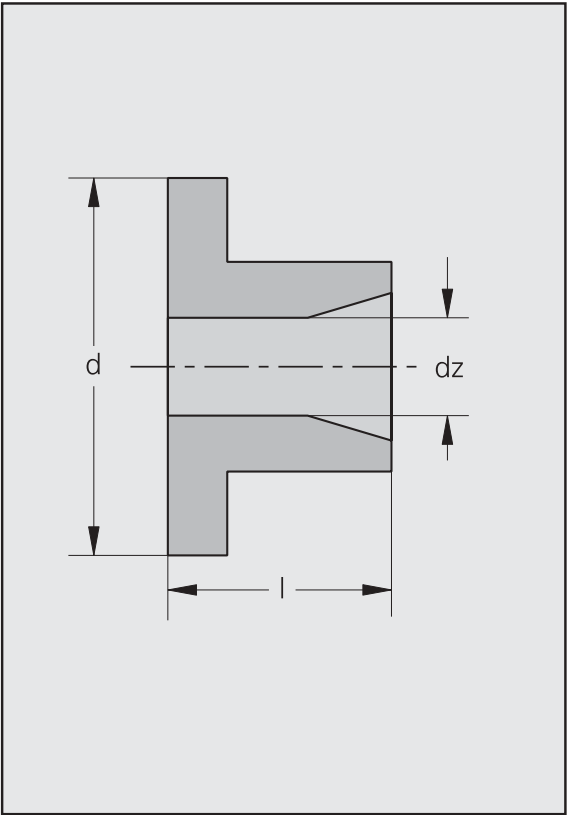
Parameters, chuck (type 1x0)

ID:	Chuck identification number
Available:	Physical availability (fixed-word list)
Jaw con.:	Code for jaw adapter
d:	Chuck diameter
l:	Chuck length
max.cl.dia. (d1):	Maximum clamping diameter
min.cl.dia. (d2):	Minimum clamping diameter
dz:	Centering diameter
max.speed:	Maximum spindle speed [rpm]

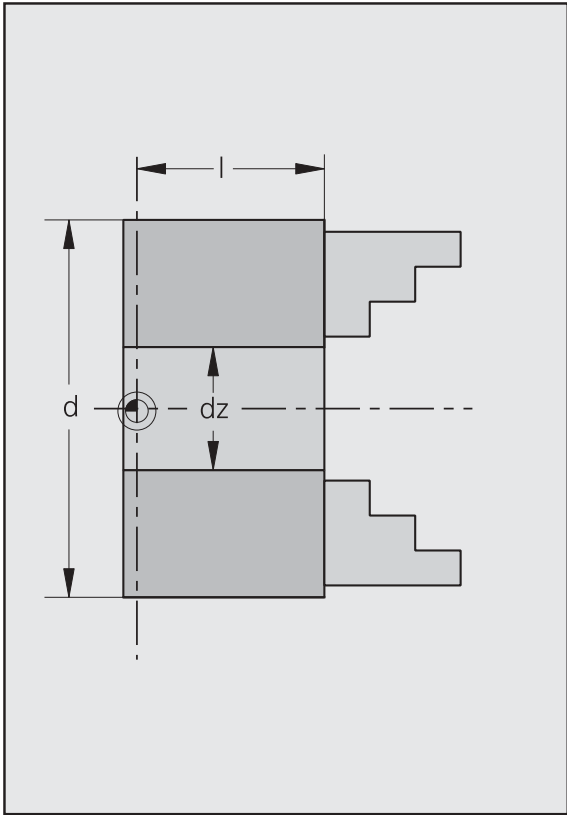
Code for jaw adapter

When specific combinations of chuck and jaws are required, enter the appropriate jaw adapter code. For both the chuck and the required chuck jaws, enter the same code.

"Jaw connection = 0": any jaw type may be used.



Collet chuck (type 110)

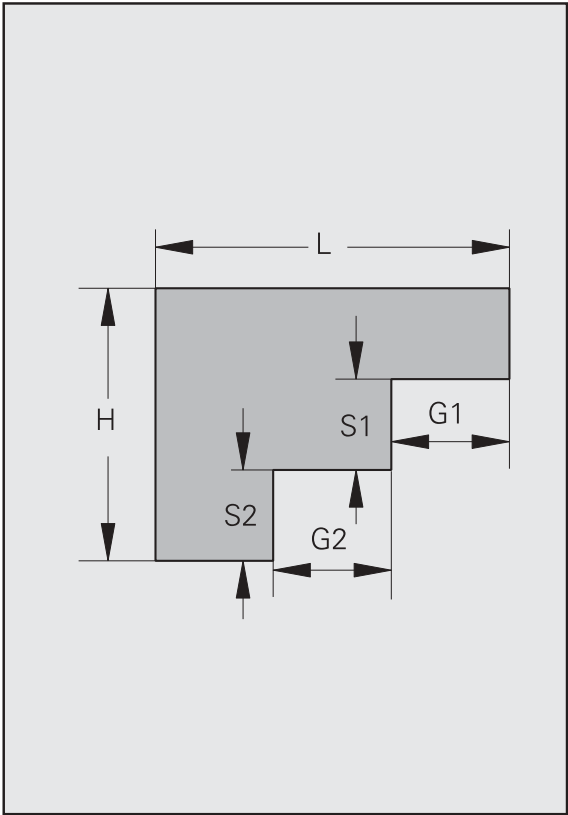


Example: Three-jaw chuck (type 130)

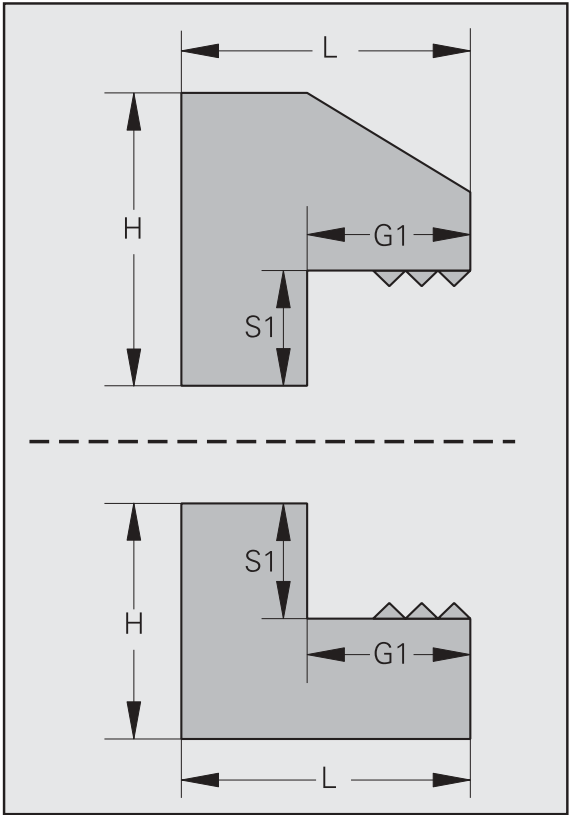
Chuck jaws

Parameters, chuck jaws (type 21x)
ID: Chuck identification number
Available: Physical availability (fixed-word list)
Jaw con.: Code for jaw adapter - must correspond to the jaw adapter code of the chuck
L: Width of jaws
H: Height of jaws
G1: Dimension, step 1 in Z direction
G2: Dimension, step 2 in Z direction

Parameters, chuck jaws (type 21x)
S1: Dimension, step 1 in X direction
S2: Dimension, step 2 in X direction
min.cl.dia.: Minimum clamping diameter
max.cl.dia.: Maximum clamping diameter



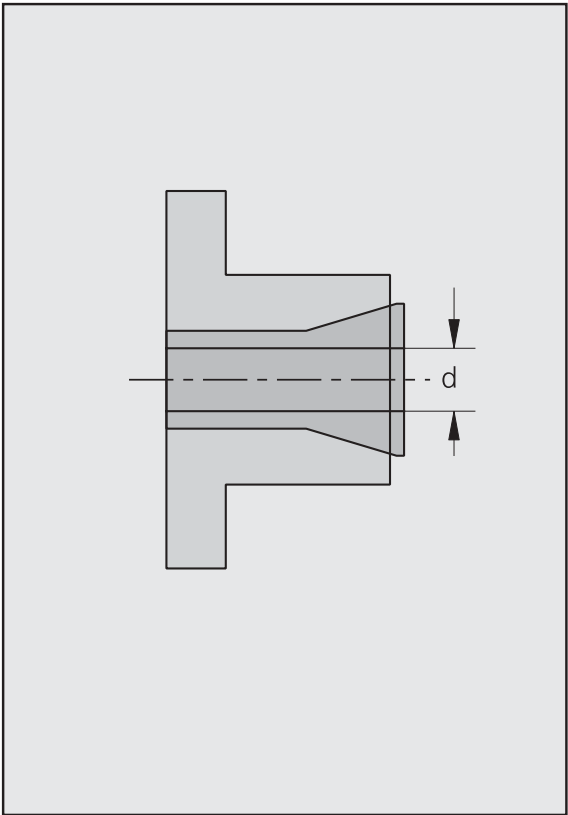
Example: Chuck jaw (type 211)



Example: Grip jaw (type 213)

Collet chuck

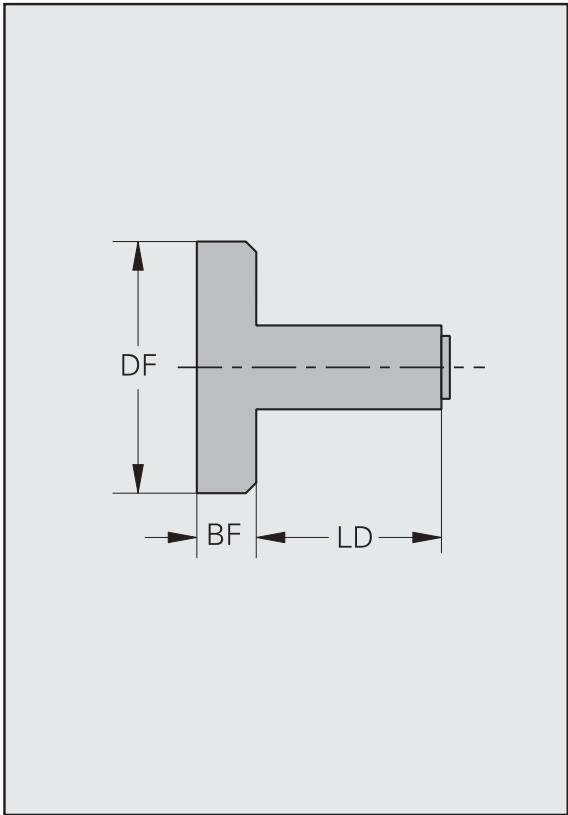
Parameters, collet chuck (type 220)
ID: Chuck identification number
Available: Physical availability (fixed-word list)
d: Diameter of collet chuck



Collet chuck (type 220)

Mandril

Parameters, mandril (type 23x)
ID: Chuck identification number
Available: Physical availability (fixed-word list)
Mandril length:
LD: Total length
DF: Flange diameter
BF: Flange width
max.cl.dia.: Maximum clamping diameter
min.cl.dia.: Minimum clamping diameter



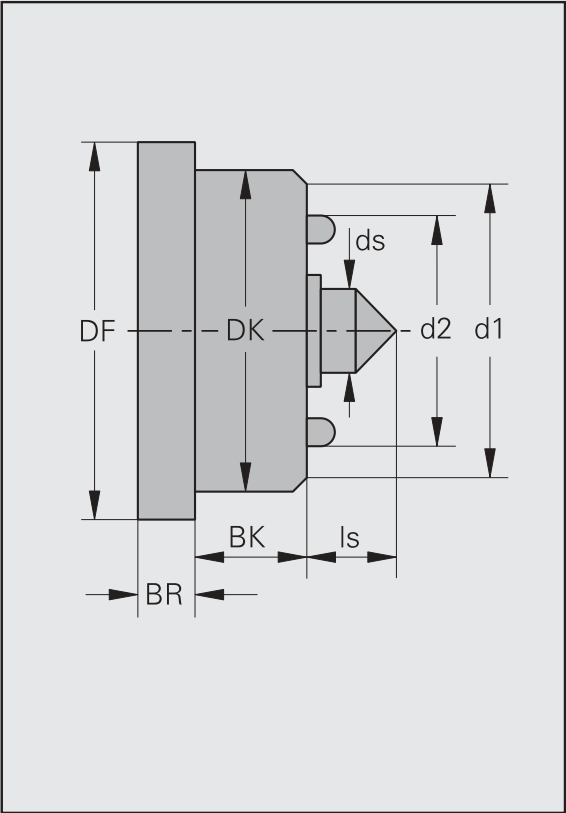
Mandril (type 23x)

Face driver

Parameters, face driver (type 24x)	
ID:	Chuck identification number
Available:	Physical availability (fixed-word list)
ds:	Tip diameter
Is:	Tip length
DK:	Body diameter
BK:	Body width
DF:	Flange diameter
BR:	Flange width
d1:	Maximum clamping diameter
d2:	Minimum clamping diameter

Rotating gripper

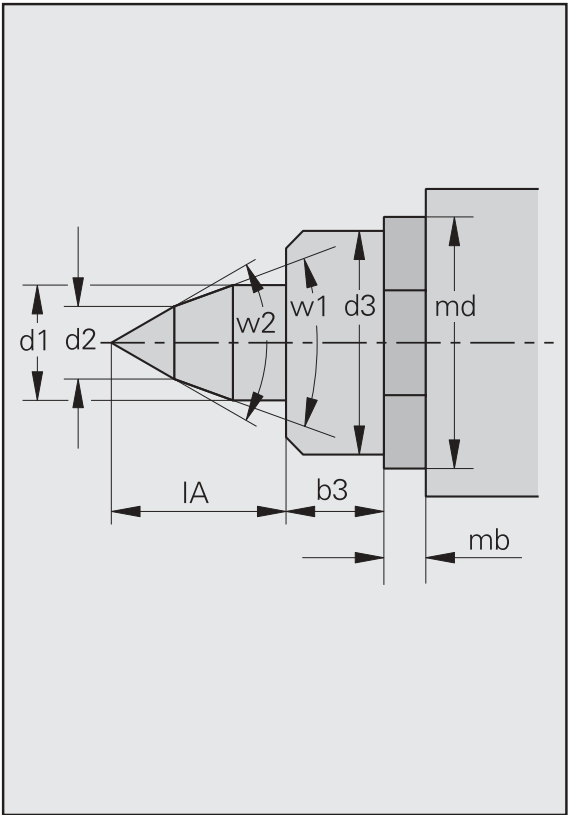
Parameters, rotating gripper (type 25x)	
ID:	Chuck identification number
Available:	Physical availability (fixed-word list)
Nom.dia.:	Diameter of rotating gripper
Length:	Length of rotating gripper
max.cl.dia.:	Maximum clamping diameter
min.cl.dia.:	Minimum clamping diameter



Face driver (type 24x)

Dead center

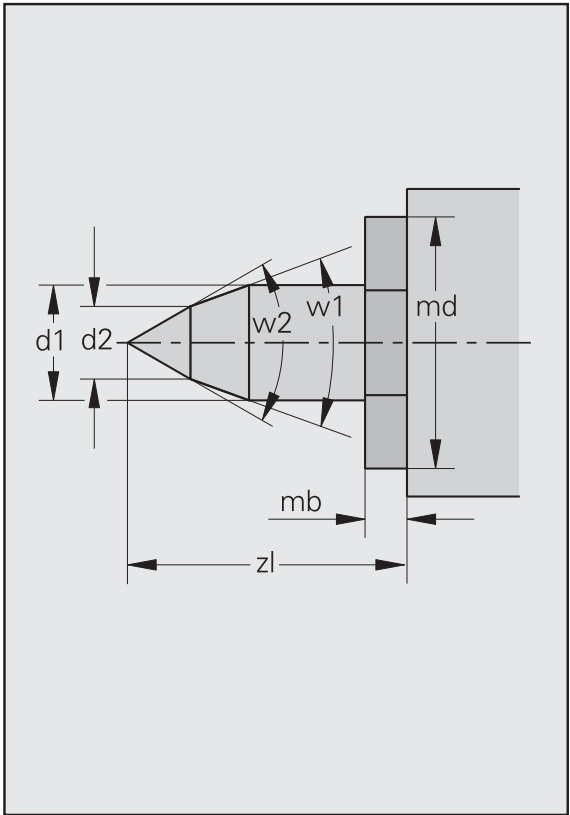
Parameters, dead center (type 26x)	
ID:	Chuck identification number
Available:	Physical availability (fixed-word list)
w1:	Lathe center angle 1
w2:	Lathe center angle 2
d1:	Diameter 1
d2:	Diameter 2
IA:	Length of conical part
d3:	Diameter of the dead center sleeve
b3:	Width of the dead center sleeve
md:	Circumference diameter of the forcing nut
mb:	Width of the forcing nut



Dead center (type 26x)

Lathe center

Parameters, lathe center (type 27x)	
ID:	Chuck identification number
Available:	Physical availability (fixed-word list)
w1:	Lathe center angle 1
w2:	Lathe center angle 2
d1:	Diameter 1
d2:	Diameter 2
zl:	Length of lathe center
md:	Circumference diameter of the forcing nut
mb:	Width of the forcing nut

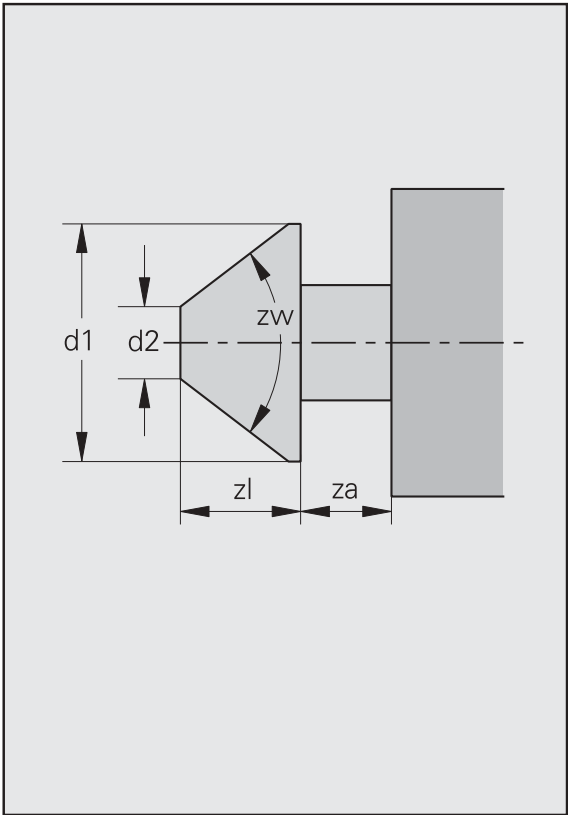


Lathe center (type 27x)

Centering cone

Parameters, centering cone (type 28x)

ID:	Chuck identification number
Available:	Physical availability (fixed-word list)
zw:	Centering cone angle
za:	Length from centering cone tip to the sleeve
d1:	(Maximum) diameter 1
d2:	(Minimum) diameter 2
zl:	Length of centering cone



Centering cone (type 28x)

8.3 Technology Database (Cutting Values)

The CNC PILOT stores the technology data according to the

- Material
- Cutting material
- Machining mode

The machining modes supported by the CNC PILOT are permanently defined and cannot be changed; materials and cutting materials can be defined in a fixed-word list.

The cutting data are managed in the **technology editor**.

Selection: "Tech(nology data)" menu item (parameter mode)

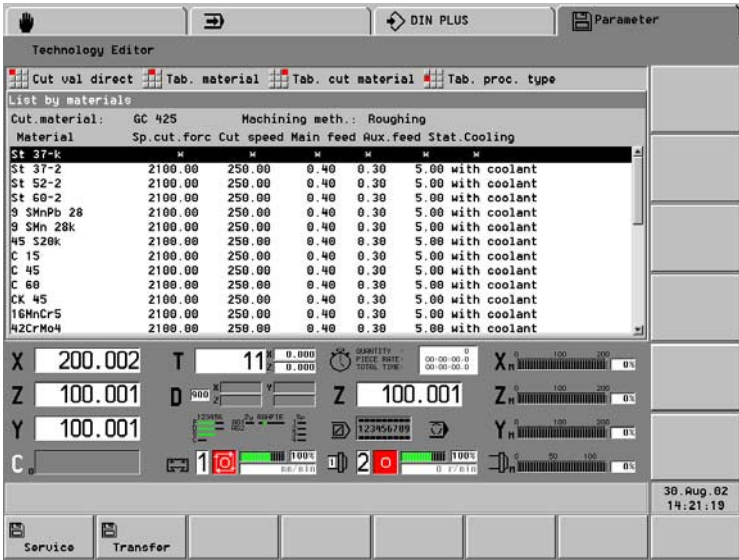
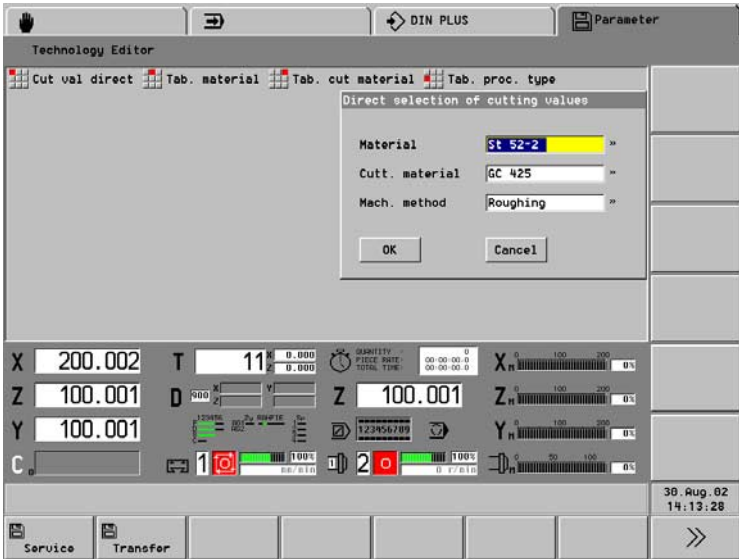
The TURN PLUS working plan generation uses the technology data. In addition, you can use the database for saving your "own" cutting data.

Cutting-value tables

- **Tab(le) material**
Define the machining mode and the cutting material – the CNC PILOT lists the cutting data according to materials.
- **Tab(le) cut. material**
Define the material and the machining mode – the CNC PILOT lists the cutting data according to cutting materials.
- **Tab(le) proc.(ess) type**
Define the material and the cutting material – the CNC PILOT lists the cutting data according to machining modes.



To enter material, cutting material and machining mode, always use the fixed-word list.



“Cut.val.direct” menu item

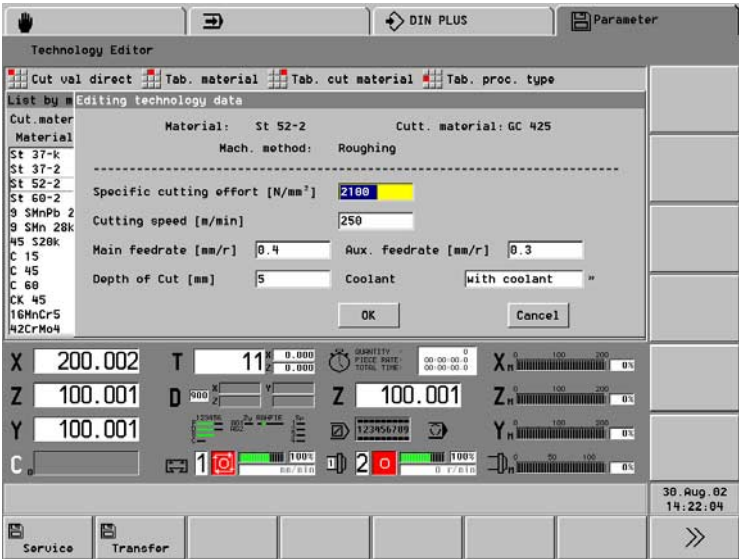
Define the material, cutting material and machining mode – the CNC PILOT opens a dialog box for editing the cutting parameters.

Cutting Parameters

- **Specific cutting force** for the material:
The parameter is for information only - it is not evaluated.
- **Cutting speed**
- **Main feed rate [mm/rev]:**
Feed rate for primary machining direction
- **Auxiliary feed rate [mm/rev]:**
Feed rate for secondary machining direction
- **Infeed (depth of cut)**
- **With/without coolant**
The automatic working plan generation (AWG) uses this parameter to determine whether coolant is used.



TURN PLUS multiplies the cutting parameters by the correction factors (CSP/ FDR/DEEP comp.) assigned to the tools (see “8.1.2Tool Data Overview”).





9

Service and Diagnosis

9.1 Service Mode of Operation

The Service mode of operation features:

- Service functions
- Diagnostic functions
- Maintenance system

Service functions: User registration and user management, language switching and different system settings

Diagnostic functions: System inspection and support when searching for errors

The **Maintenance system** reminds the machine user about necessary maintenance tasks.

9.2 Service Functions

9.2.1 Access Authorization

Functions such as editing important parameters are reserved for privileged users.

The CNC PILOT permits access when the correct **password** is entered. The access authorization remains in effect until the user logs off or until a second user logs on correctly.

Each user is assigned a "password" consisting of a four-digit number – it is entered "masked" (not visible).

The CNC PILOT differentiates between:

- No protection class
- NC programmer
- System manager
- Service personnel (of the machine manufacturer)

"Log-on" menu item

To log on, select your name from the list of users and enter your password.

"Log-off" menu item

The CNC PILOT does not support automatic log-off. User log-off is necessary in order to protect your system from unauthorized access.

"User srv." (User service) drop-down menu

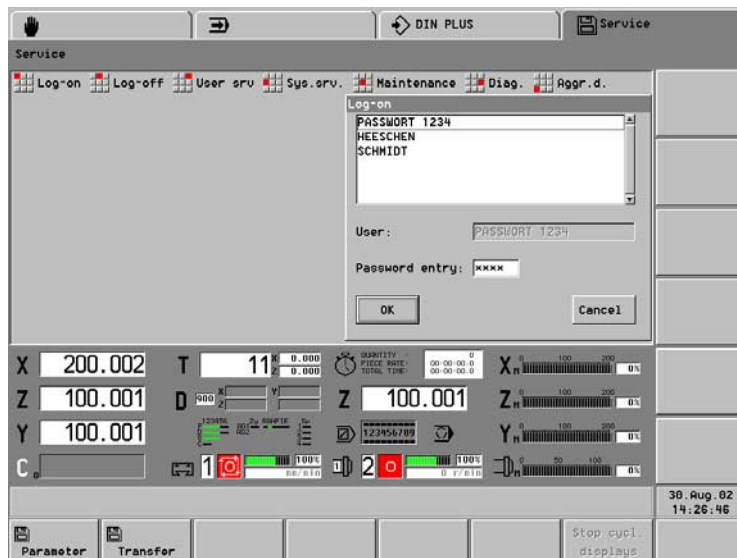
User service functions are only available after logon as a system manager.

■ Enter user

Enter the name of the new user, assign a password and user class. Precondition: You are logged on as a system manager.



Some service and diagnostic functions are not accessible (reserved for service and commissioning personnel).



■ **Cancel user**

Select the user to be deleted and confirm with OK.

■ **Change password**

Every user can change his or her assigned password. To safeguard against misuse, the user must first enter the "old" password before assigning a new one.

■ **"Maintenance" menu item**

See "9.3 Maintenance System"



■ The CNC PILOT is shipped with the user name "Password 1234" and the password "1234" (user class: system manager). To enter further users, log on with the user name "Password 1234." You should then delete the user "password 1234."

■ The CNC PILOT prevents that the last system manager is deleted. Make sure, however, to memorize your password.

9.2.2 System Service

■ **"Sys.srv." (System service) drop-down menu**

■ **Date/Time**

Error messages are recorded together with the date and time they occurred. Since all errors are stored in a log file for a long period of time, you should always ensure that the date and time are correctly set. These data facilitate the fault diagnosis in case you require field service.

■ **Language switching**

Select the language with the soft key ">>" and press "OK". The selected screen language becomes effective as soon as you restart the CNC PILOT.

■ **FWL-editing – Language dependent** – currently not used.

■ **FWL-editing – Language independent:**

- Material (file name: "OTEMATER")
 - Cutting material (file name: "OTESTOFF")
 - Fittings (file name: "0WZPASSU")
 - "0Listbox": Currently not used
- (FWL = fixed word lists – see "9.2.3 Fixed Word Lists")

■ **Aux. images ON/OFF**

When "Aux. images ON" is active, the graphic support windows of Machine mode are not displayed.

■ **Editing switch ON/OFF**

This function protects the operating modes

- DIN PLUS
- TURN PLUS
- Parameter

from unauthorized access. When "Editing switch ON" is active, these menu items can only be selected if you are logged on as an NC programmer (or higher).

■ **"Aggr.d." (Component ("Aggregate") diagnosis) drop-down menu**

Using the submenu items, you can call diagnosis functions defined by the machine manufacturer (see Machine Manual).

9.2.3 Fixed-Word Lists

Materials and cutting materials

The terms for materials and cutting materials from the technology database are contained in fixed-word lists. Use them to adapt this database to your requirements (see also "7.5 Cutting-Value Database").

Fits

The parameter "Fit" is included for the delta drill and reamer tools. Specify the desired fit qualities in the "0WZPASSU" fixed word list.

Note on editing fixed-word lists:

- Maximal 64 entries
- **Code**
 - Number from 0..63
 - Assign each code only once
- **Name**
 - Maximum 16 characters

Editing a fixed-word list

Select "Sys.srv. – FWL-editing – Language independent."

Select:

- "0TEMATER" (material)
- "0TESTOFF" (cutting material)
- "0WZPASSU" (fit quality)

Editing an entry

Select the position to be edited; press ENTER.

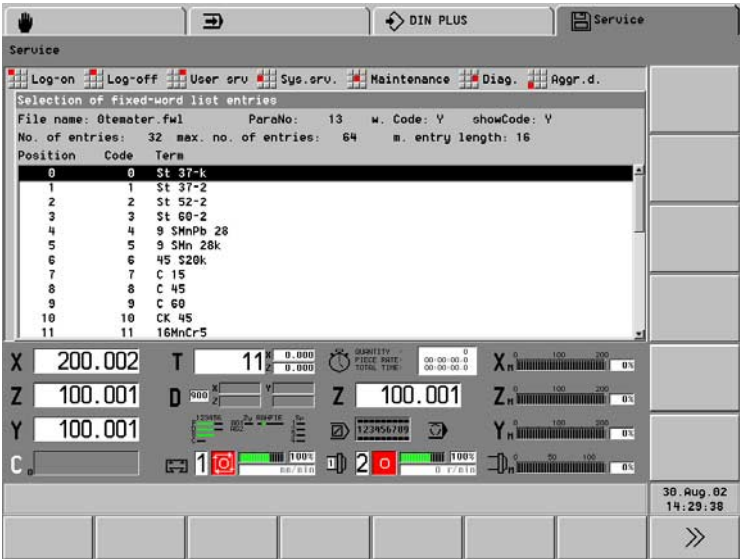
Edit the "Code" or "Term" and press OK. The CNC PILOT saves the data.

New entry

Ins

Opens the "Editing fixed-word lists" dialog box.

Enter the "Code", "Term" and press OK. The CNC PILOT saves the data.



9.3 Maintenance System

Maintenance System

The CNC PILOT reminds the machine user of the required maintenance and repair tasks. Every action is described briefly (assembly, maintenance interval, responsible person, etc.). This information is displayed in the "Maintenance and repair actions" list. A comprehensive description of the maintenance action is provided if desired.

After the user completes a maintenance action, he acknowledges it. Then the maintenance interval begins again. The CNC PILOT saves the time of acknowledgment together with the nominal date in a log file. The acknowledgment log files can be called by the service personnel and evaluated. You can see at least the last 10 acknowledgments.

Display of maintenance status: "Traffic light" at right next to the date/time information

- Green: No maintenance actions required
- Yellow: At least one maintenance action is due soon
- Red: At least one maintenance action is due or overdue

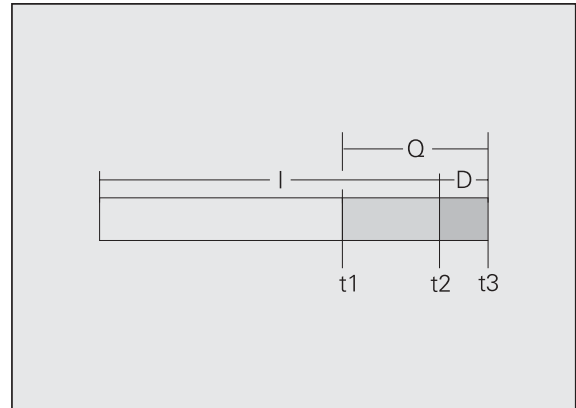
The status with the highest priority is displayed (red before yellow, yellow before green).

Dates and intervals (see illustration):

- **I – Interval:** The period of time between maintenance actions as fixed by the machine tool builder.
During the on time of the control, the current maintenance interval is continuously reduced. The remaining time appears in the "When" column.
- **D – duration:** Period defined by the machine tool builder between "due" and "overdue" maintenance actions.
- **Q – acknowledgment period:** Within this period, the maintenance action can be conducted and acknowledged.
- **t1 – maintenance action is due soon:**
 - Starting from this time point the maintenance action **can** be completed and acknowledged.
 - The status is marked yellow.
 - Calculation: $t1 = \text{Early warning entry} * \text{interval} / 100$
- **t2 – maintenance action is due:**
 - Starting from this time point, the maintenance action **should** be completed and acknowledged.
 - The status is marked red.
 - Calculation: $t2 = \text{interval}$
- **t3 – maintenance action is overdue:**
 - The time point of the maintenance action has been **exceeded**.
 - The status remains red.
 - Calculation: $t3 = \text{interval} + \text{duration}$



- **Prerequisite:** The machine tool builder must enter the required actions and provide complete descriptions of them.
- All status changes including acknowledgment of the maintenance actions are reported to the PLC. Refer to the machine manual to see whether further consequences must be made from due or overdue maintenance actions.



Explanation:

I: Interval

D: Duration

Q: Acknowledgment time period

t1: Maintenance action will soon be due


t2: Maintenance action is due

t3: Maintenance action is overdue

List of "maintenance actions"

- **Type:** See "type of maintenance action" table
The **status** is distinguished by background color:
 - No color: No maintenance action required
 - Yellow: Maintenance action is due soon
 - Red: Maintenance action is due or overdue
- **Location:** Location of the assembly
- **Assembly:** Designation of the assembly
- **When:** Time remaining until maintenance action is due (= remaining time of the maintenance interval)
- **Duration:** Time period between due and overdue maintenance actions
- **Who:** Person responsible for performing the action
- **Interval:** Duration of the maintenance interval
- **Early warning:** Defines the time point of the "action is soon due" status (relative to the maintenance interval)
- **Documentation reference and type:**
 - Entry exists: the "Info actions" soft key calls a comprehensive description of the maintenance action
 - No entry: There is no comprehensive description of the maintenance action available.

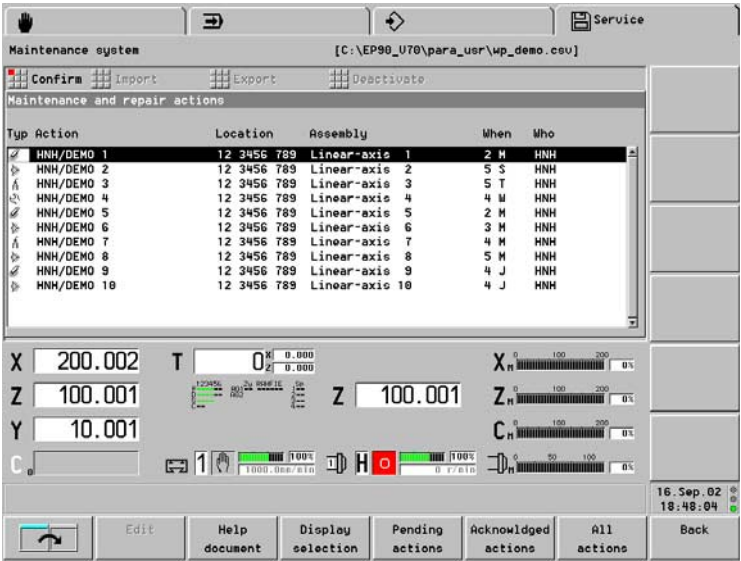
Calling the maintenance system: "Maintenance" menu item (Service operating mode)

 Return to "Service"

After calling the maintenance system, the **"Maintenance and repair actions" list** are displayed with all actions. The information is divided into Parts 1 and 2 (switch by soft key).

Operation

- Vertical arrow keys; Page Up/Dn: move the cursor within the list of actions.
- Enter: Opens a dialog box with the parameters of the action marked by the cursor



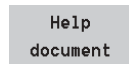
"Maintenance system – general" soft keys



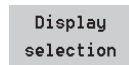
Display "Part 2" of the actions list



Display "Part 1" of the actions list



Call comprehensive description of the action



Switch to the soft-key row "Type/status of actions"



Switch back to the "maintenance system" soft-key row

Type of maintenance action



Cleaning



Inspection



Maintenance



Repair







"—" before the symbol: The maintenance system is deactivated

Selecting the list



You can open the list "Maintenance and Repair Tasks" according to the following criteria:

All actions	List of all maintenance tasks
Pending actions	List of the "pending, current and overdue maintenance tasks"
Display selection	Switch the soft-key row to "Type/Status of Tasks"

Type of tasks:

	List of repair tasks
	List of maintenance tasks
	List of inspection tasks
	List of cleaning tasks

Status of the tasks:

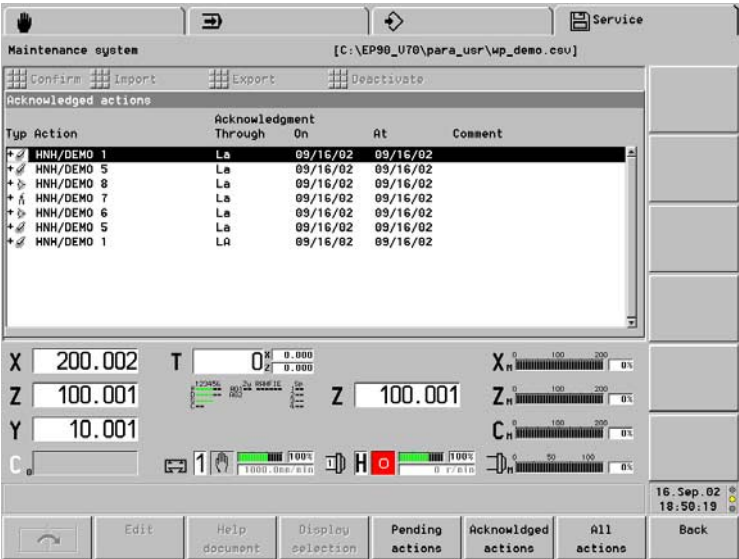
	List of the "current and overdue maintenance tasks"
	List of the "pending maintenance tasks"

Acknowledged measures

Acknowledged actions	Lists the acknowledged maintenance measures
----------------------	---

List of acknowledged measures:

- **Type:**
 - Symbol: See "Type of maintenance actions" table
 - "+": Measure was acknowledged
- **Measure:** Designation of the maintenance action
- **Acknowledged – through:** Name of the acknowledger
- **Acknowledgment – on:** Date of the acknowledgment
- **At:** Time of "maintenance action is due" (t2)
- **Comment** of the acknowledger



Time data (German / English)

M / M:	Minutes
S / H:	Hours
T / D:	Days
W / W:	Weeks
J / Y:	Years

Parts of a time unit are given as decimal numbers. Example: 1.5 S = 1 hour and 30 minutes.

9.4 Diagnosis

Call: "Diag.(nosis)" drop-down menu in the "Service" mode

Esc Return to "Service"

The "Diagnosis" submenu provides information, test and control functions that help you with troubleshooting.

"Info" menu item

Provides information on the software modules being used.

"Display" drop-down menu

- **Memory** – only accessible by the service personnel
- **Variables** displays the current contents of approx. 500 V variable (see also "4. 15.2V Variables").
 - "—": The variable is not initialized
 - "???": The variable is not available
- **Input/Output** displays the current status of all input/output positions (interface between CNC PILOT and lathe).
- **16 Inputs/Outputs:** Enter up to 16 inputs/outputs in the "Select I/O's for display" dialog box. After the dialog box has been concluded, the CNC PILOT displays the current status of these inputs/outputs. Each change of status is displayed immediately.
To exit the display function, press the ESC key.
- **Cyclic memory** – is reserved for the service personnel
- **Variables cyclic:** Enter a V variable. The CNC PILOT displays the current value. Each change in value is displayed immediately.
- **Inputs/Outputs cyclic:** Enter an I/O position. The CNC PILOT displays the current status. Each change of status is displayed immediately.



The cyclic display window is superimposed on part of the machine display. To cancel cyclic displays, select "Display - Stop cyclic displays."

“Log files” drop-down menu

Errors, system events, and data exchange between different system components are recorded in log files. Some log files are save “on command” and can be used by the service personnel for troubleshooting.

- The **display error log** function displays the most recent error message. To view further entries, press the PgUp/PgDn keys.
- **Save error log file** makes a copy of the error log file (file name: error.log; Directory: Para_Usr). Existing “error.log” files are overwritten.
- **Save Ipo trace** saves information on the most recent interpolator functions (file names: IPOMakro.cxx, IPOBewbe.cxx, IPOAxCMD.cxx—xx: 00..99; directory: data).

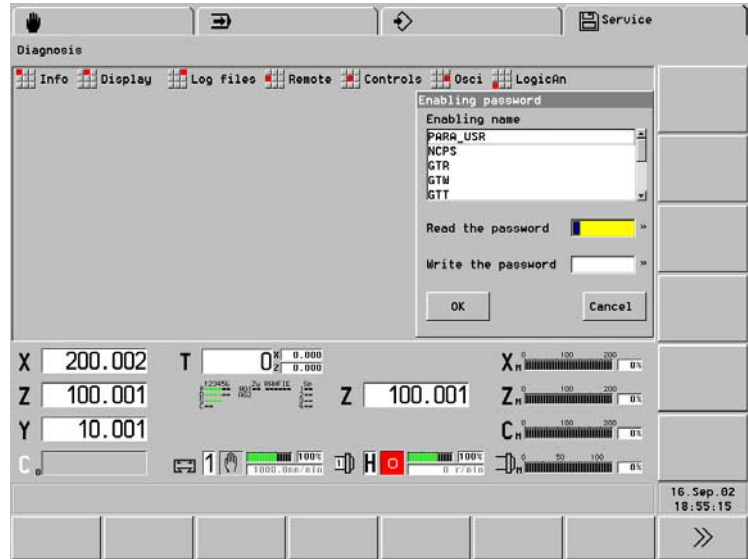
“Remote” drop-down menu

The remote functions support **remote diagnosis**. For information on the “Remote” functions, refer to your machine manufacturer.

“Controls” drop-down menu

- The **hardware—System info** option supplies you with information on the hardware components used.
- **Options:** You receive an overview of the available and installed options of the CNC PILOT (see also “1.3 Expansion Stages (Options)” and Control Parameters 197).
- **Network—Settings:** This menu item calls the Windows “Network” dialog box. The CNC PILOT is entered as a “Client for Microsoft Networks.” Details on installing and configuring networks are available in the documentation or in the on-line help function of Windows.
- **Network—shared password:** Assign individual passwords for read and write access. However, the passwords are valid for all “shared directories” (see also “10.3.1 Releases, File Types”).
The “Share names” listed in the “shared password” dialog box are for your information. Entries can only be made in the “Read password” and “Write password” input fields. The password is entered “masked” (not visible).
- **Network—Network ON:**
- **Network—Network OFF:** Switches the network adapter of the control on or off, then restart the system to put it into effect.

The menu items “Osci(loscope), LogicAn(alyzer)” are reserved for service personnel





10

Transfer

10.1 The Transfer Mode of Operation

The Transfer mode is used for **data backup** and **data exchange** with other IT systems. It moves files from one location to another. The files contain NC programs (DIN PLUS or TURN PLUS programs), parameter files, or files with information for service personnel (oscilloscope data, log files, etc.)

The Transfer mode also includes **organizational functions** such as copying, deletion, renaming, etc.

Data Backup

HEIDENHAIN recommends saving your CNC PILOT programs on a PC in regular intervals.

You should also back up the parameters. Since the parameters are not changed very often, however, you only need to back up the parameters from time to time, as required. See "10.4.2 Backup Up Parameters and Operating Resource Files."

Data exchange with DataPilot

HEIDENHAIN offers the PC program package **DataPilot 4290** to complement the CNC PILOT machine control. DataPilot has the same programming and testing function as the control. This means that can create TURN PLUS and DIN PLUS programs on the PC, test it with program simulation and transfer it to the machine control.

Systems for data backup/data exchange

The DataPilot also includes backup functions. Of course, you can also use alternative Windows operating system functions or other PC programs available on the market for data backup.

Printer

With the **organization functions** you can printout DIN PLUS programs and parameter/operating resource data. TURN PLUS programs cannot be printed out.

The CNC PILOT prepares for print job for A4 format (21 cm x 29.7 cm).



Menu for transfer mode



Network – activates the Windows network and shows the "masked" files of the CNC PILOT and the communication partner.



Serial – activates serial data transmission and displays the "masked" files of the CNC PILOT.



FTP – activates the FTP network and shows the "masked" files of the CNC PILOT and the communication partner.



Calling the **Organization** (local)



Parameter conv(ersion) – Converts parameters/operating resources from "internal format" to ASCII format – or vice versa; prepares data backup – downloads saved data



Setting the network, FTP, serial interfaces or printer parameters



- Files in TURN PLUS format are processed only by CNC PILOT or DataPilot – they are not readable.
- Service files support troubleshooting. Theses files are normally transferred and evaluated by service personnel.

10.2 Transfer Systems

10.2.1 General Information

Interfaces

HEIDENHAIN recommends data transfer over an **Ethernet-interface**. That guarantees high transmission speed, high reliability, and convenient operation. Data transmission via **serial interface** is also possible.

■ Windows networks (Ethernet interface):

With a Windows network you can integrate your lathe in an LAN network. The CNC PILOT supports the usual networks under Windows. The CNC PILOT allows you to send/receive files. Other computer systems integrated in the network have access to read from and write to shared directories, independent of the CNC PILOT's activities.

The CNC PILOT usually logs on in the network during startup and does not log off until the system is terminated.

■ FTP (File Transfer Protocol) (Ethernet interface):

With "FTP" you integrate your lathe in a LAN network. To do this, an FTP server must be installed on the host computer (with Windows NT and UNIX, the FTP server is included in the operating system; an FTP server is available for Windows 95/98). The CNC PILOT allows you to send/receive files.

The CNC PILOT does **not** support server functions. This means that other computer systems in the network cannot access CNC PILOT files.

■ Serial

You transfer programs or program files via serial interface – **without protocol**. Ensure that your communications partner complies with the defined interface parameters (baud rate, word length, etc.).

■ Network printer

the CNC PILOT transmits print jobs to the defined standard printer. Prerequisites:

- Installed printer driver
- Declaration as standard printer
- Device name: **STD** ("Printer settings" dialog box)

■ Local printer

The CNC PILOT transmits print jobs to the COMx interface (entry in the "Device name" field of the "Printer settings" dialog box).



HEIDENHAIN recommends having trained personnel put your printer into service.

10.2.2 Configuring for Data Transfer

Configuring a network

Windows networks and FTP networks are configured in the Windows operating system.

- ▶ Log-in as "system manager."
- ▶ Select "Controls – Network – Settings." This menu item calls the Windows dialog box "Network."
- ▶ Perform network configuration. The CNC PILOT is entered as "Client for Microsoft Networks." Details on installing and configuring networks are available in the documentation or in the on-line help function of Windows.



HEIDENHAIN recommends configuration of the Windows networks by the authorized personnel of your machine manufacturer.

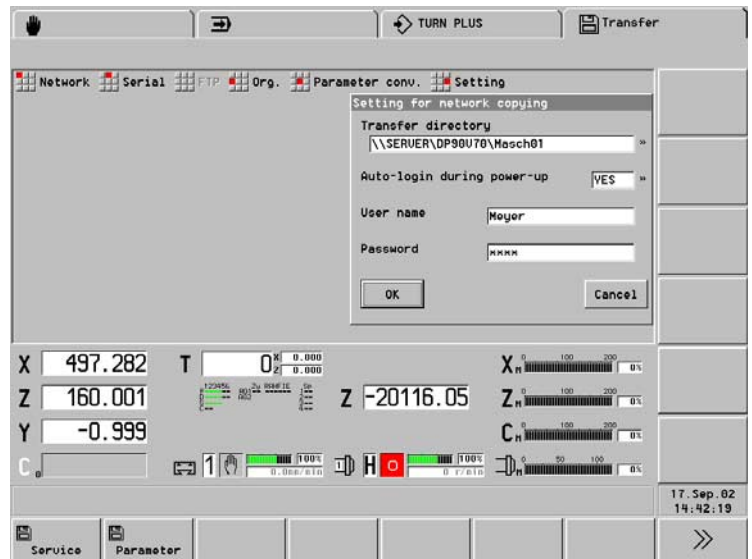
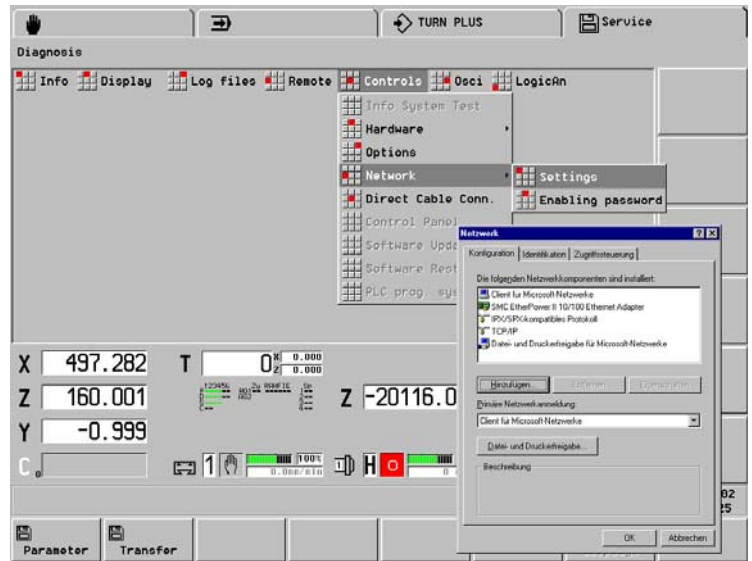
Settings for Windows network

- ▶ Log-in as "system manager"
- ▶ Select "Setting – Network" (Transfer mode)
- ▶ "Setting for network copying" dialog box:
 - Transfer directory: Enter the path of the communication partner (see next page)
 - Auto-login during power-up:
 - YES: The CNC PILOT accepts the login with the data entered in "User name" and "Password"
 - NO: You enter the user name and password during system start

Recommendation: Use the automatic login

Activate Windows network:

- ▶ Select "Network" (in Transfer mode) –With the defined mask, the CNC PILOT displays:
 - Files of its own system
 - Files of the defined transfer directory (Files of the communication partner)



Transfer directory

You enter the computer name, enabling name, and path of the communication partner in "transfer directory" ("Setting - Directory" dialog box) in the following form:

\\Computer name\Enabling name\Path

Example:

\\DATAPILOT\C\DP90V70\MASCH\MASCHINE1

You define the "computer name" and "enabling name" on the PC of the communication partner. In this example, the "C" disk is enabled.

Whether you define the complete path or parts of the path depends on your organization.



Define the "transfer directory" without subdirectory. The CNC PILOT adds the last stage, depending on the file type.

Settings for FTP

- ▶ Log-in as "system manager"
- ▶ Define in control parameter 11 ("FTP parameter"):
 - Use FTP: 1 (=YES)
- ▶ Select "Setting – FTP" (Transfer mode)
- ▶ "Setting FTP" dialog box:
 - User name, password: for login at the host computer
 - Address/Name FTP server: Enter the server name or IP address of the host computer



The menu item "FTP" and "Setting – FTP" are selectable only if "Use FTP = Yes" is entered in control parameter 11.

Activate FTP:

- ▶ Select "FTP" (in Transfer mode) –With the defined mask, the CNC PILOT displays:
 - Files of its own system
 - Files of the defined transfer directory (Files of the communication partner)



Configure serial interface or printer

- ▶ Login as "system manager"
- ▶ Select "Setting – Serial/Printer" (in the Transfer mode)
- ▶ Enter data in "Setting serial/printer" dialog box

Parameter

Define the following interface parameters in consultation with your communications partner.

- **Baud rate** (in bits per second): The baud rate is defined according to the local condition (cable length, interference, etc.). A high baud rate has the advantage of fast data transfer. A lower baud rate, however, is more stable.
- **Word length:** Choose between 7 or 8 bits per character.
- **Parity:** If you select even/odd parity, the CNC PILOT adds a parity bit so that an even/odd number of set bits are transferred per character. The parity can be checked at the distant terminal. If you define "No parity," the characters are transferred as they are saved. The parity bit is transmitted in addition to the number of bits defined in "Word length."
- **Stop bits:** Choose between 1, 1 1/2 and 2 stop bits.
- **Protocol:**
 - **Hardware** (hardware handshake): The receiving terminal informs the transmitting terminal by way of the RTS/CTS signals that it is temporarily unable to receive data. A hardware handshake presupposes that the RTS/CTS signals are hardwired in the data transfer cable.
 - **XON/XOFF** (software handshake): The receiving terminal transmits XOFF if it is temporarily unable to receive data. with XON is indicates that it can receive data again. The software handshake needs no RTS/CTS signals in the transmission cable.
 - **ON/XOFF** (software handshake): The receiver transmits XON at the beginning of data transmission to indicate that it is ready to receive. When the receiver is temporarily not able to receive data, it transmits XOFF. With XON is signalizes that it can receive more data. The software handshake does not require the transmission of RTS/CTS signals over the data transfer cable.
- **Device name:**
COM1: RS-232-C/V.24 data interface



- The menu items "Serial" and "Setting – Serial" are selectable only if an interface is assigned in "External input/output" (control parameter 40).
- The parameters of the serial interface are saved in one of the control parameters 41 to 47 (depending on the setting in control parameter 40).

10.3 Data Transfer

10.3.1 Enabling, DataTypes

Enabling – CNC PILOT

See "Enabled directories"

You can protect these directories from access for reading and/or writing by assigning **passwords** (menu item "Controls – Network – Enabling password" in the Service/Diagnosis operating mode – see "9.3 Diagnosis")

If you enter no password, all communication partners have access to the directories.

Enabling – communication partners

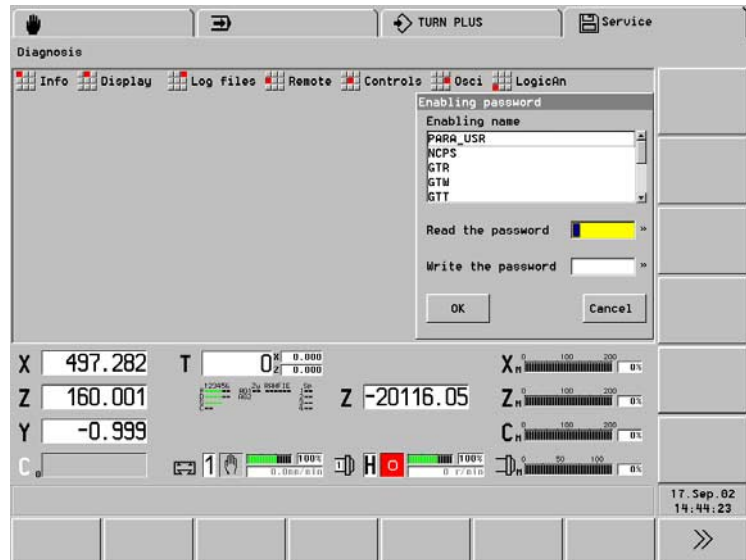
The communication partner can assign passwords for reading or writing access (Windows: "System control – Networks – Access control to shared levels"). In this case, the Windows dialog box "Enter network password" appears when you try to access directories of the remote system.

If only **one password** is used, it can be saved. As a result, the dialog box appears only once (and each time the password has been changed). The password saved is checked each time you try to access further directories. If the password for read permission differs from the one for write permission, the "Enter network password" dialog box appears each time you try to access a directory after having restarted the CNC PILOT.

File types

Select the desired file type in the dialog box "Mask of files":

- All NC programs (DIN PLUS programs)
- NC main programs (DIN PLUS programs)
- NC subprograms (DIN PLUS programs)
- Template files (DIN PLUS templates)
- TURN PLUS complete (workpiece blank and finished part description and work plan)
- TURN PLUS workpieces (workpiece blank, finished part description)
- TURN PLUS workpiece blanks (blank description)
- TURN PLUS finished parts (finished part description)
- TURN PLUS contour trains (description of contour trains)
- TURN PLUS Turr.(et) lists
- TURN PLUS Mach.(ining) seq.(uence)
- Parameter files (PARAM directory)
- Parameter backup (BACKUP directory)
- Program head lists (subfiles for program head entries)



Danger of collision!

Other computer systems in the network may overwrite CNC PILOT programs. When organizing the network and granting access rights, ensure that only authorized persons have access to the CNC PILOT.

Shared directories in CNC PILOT

...**NCPS**: NC main programs and subprograms, template files

...**VGTR**: Workpiece blank descriptions (TURN PLUS)

...**VGTF**: Finished part descriptions (TURN PLUS)

...**GTW**: Workpiece descriptions (TURN PLUS)

...**GTC**: Complete programs (TURN PLUS)

...**GTT**: Contour train descriptions (TURN PLUS)

...**GTL**: Turret lists (TURN PLUS)

...**GTB**: Machining sequences (TURN PLUS)

...**PARAM_USR**:

- Subfiles for program head entries
- Converted parameter and operating resource files
- (Saved) error log file

...**DATA**: Files for the service personnel

...**BACKUP**: Files for data backup (backup/restore)


10.3.2 Transmitting and Receiving Files


After selecting Transfer mode, select by menu the transmission method:

- **Network:** Windows networks
- **Serial:** Serial data transfer
- **FTP:** File Transfer Protocol

Displays

- Left window: File belonging to CNC PILOT
- Right window:
 - Network and FTP: Directory of the communications partner
 - Serial transfer: Defined interface

 Back to transfer menu




- If after a certain waiting time the communications partner cannot be reached, an error message appears.
- Parameters and operating resource data must be converted before transmission and after reception (see "10.4.1 Parameter and Operating Resource Transfer")

Changing communication partners

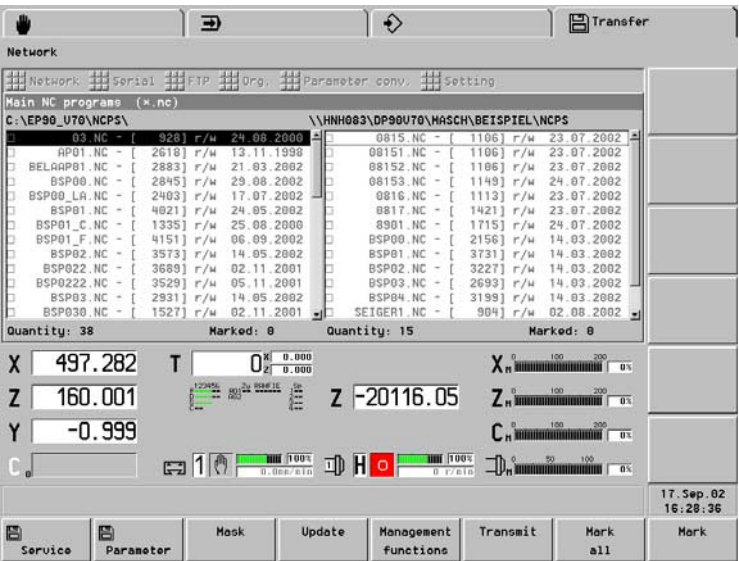
Change the entry in "Transfer directory" or in "Address/Name of FTP server" ("Settings – ." dialog box).





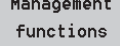

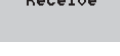


Changing the file group, changing "mask"

The current mask setting is displayed below the menu line.

Mask

- **Data type:** See "10.3.1 Enabling, File Types"
- **Sorting:** Sort files sorted by name or date
- **Mask:** Only those entries are displayed that the mask allows. Wildcards:
 - *: Represents any characters at this position.
 - ?: Represents any one character at this position.The CNC PILOT automatically adds "*" to any entered mask.



Soft keys	
 Service	Change to Service mode
 Parameter	Change to Parameter mode
 Mask	Define file type, sorting and masking
 Update	Update the file list
 Management functions	Call the "Organizations functions" – see "10.4 File Organization"
 Transmit	Transmit the marked files
 Receive	Download the marked files from the communications partner – for serial transfer: The CNC PILOT goes to readiness to receive
 Mark all	Mark all files
 Mark	Mark a file


Continued ►

Operation

- Vertical arrow keys; Page Up/Dn: move the cursor within the list of files.
- Arrow left/right: Switches between the left and right window – while also switching the CNC PILOT between readiness to transmit and receive
- Enter character/character series: The cursor positions to the next file name beginning with this character sequence
- Enter (with DIN PLUS programs, parameters and operating resource files): Shows the file content. You close the file by pressing Enter again (or Esc).

Mark all	Marks all displayed files – pressing again unmarks them
Mark	This soft key or “+” (plus key) marks the selected file – pressing again unmarks it
Transmit	<ul style="list-style-type: none">■ Network or FTP: Marked files from the CNC PILOT to the communications partner. If the file already exists, you must answer the inquiry “Overwrite?”■ SerialTransfer: Marked files are transmitted.
Receive	<ul style="list-style-type: none">■ Network or FTP: Marked files are transferred from the communications partner to the CNC PILOT. If the file already exists, you must answer the inquiry “Overwrite?”■ SerialTransfer: The CNC PILOT switches to readiness to receive or receives data. If the file already exists, you must answer the inquiry “Overwrite?”


- **Mouse operation:** You can use a mouse to position the cursor, and mark and open a file (with DIN PLUS programs, parameter files and operating resource files).

 During serial transfer, first start the receiver and then the transmitter.

10.4 Parameters and Operating Resources

10.4.1 Converting Parameters and Operating Resources

Call: Menu item "Parameter conv.(ersion) – Save / Load"

 Back to Transfer menu

The CNC PILOT saves parameters and operating resource data in internal formats and in special directories in CNC PILOT. Before transmission the data are converted to ASCII format and transferred to the PARA_USR directory.

Inversely, received parameters and operating resource files are saved in the PARA_USR directory. In a further step, you "activate" these files. This means that the data are converted to internal format and transferred to special directories in the CNC PILOT. After this step, the CNC PILOT works with the parameter and operating resource data received.

During conversion of parameters/operating resources you define the name of the backup file and influence its output as follows (in the "Save parameters" dialog box):

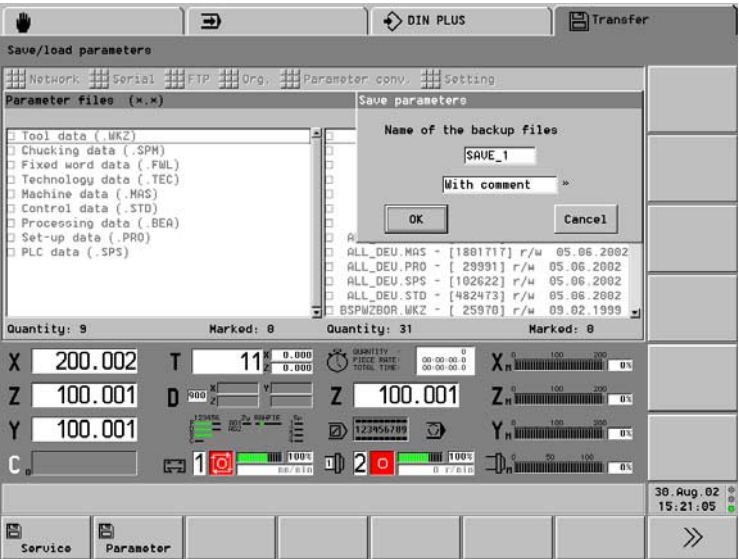
- **No comment:** Only parameter and operating resource data are output
- **With comment:** Explanatory comments are output along with the parameter/operating resource data

Changing the mask (only in the right window)

The current mask setting is displayed below the menu line.

 Mask

- **Sort:** Sort files by name or date
- **Mask:** Only those entries are displayed that the mask allows. Wildcards:
 - *: Represents any characters at this position.
 - ?: Represents any one character at this position.The CNC PILOT automatically adds "*" to any entered mask.



Parameter-conversion soft keys

Selective collection	Convert individual parameters/operating resource data
Save parameters	Convert parameters/operating resources
Load parameters	"Activate" marked files
Management functions	Call the organization functions – see "10.4 File Organization"



To parameter and operating resource files can be transmitted by serial interface (7-bit transfer), some special characters are replaced in the comments by "_".

Continued ►

Operation

- Vertical arrow keys; Page Up/Dn: move the cursor within the list of files.
- Horizontal arrow keys: move from left to right window and vice versa.
- Enter (only in right window): displays the file content – you close the file by pressing Enter again (or Esc key)

Selective collection

Opens the selected parameter/operating resource file and presents the individual parameter/operating resource file for marking and subsequent transfer.

Save parameters

Converts and transfers the marked parameter/operating resource file or the marked parameters/operating resources (selection) in the PARA_USR directory.

Load parameters

Calls the marked parameter/operating resource file from the PARA_USR directory, converts the data to the internal format and overwrites the existing parameter/operating resource data.

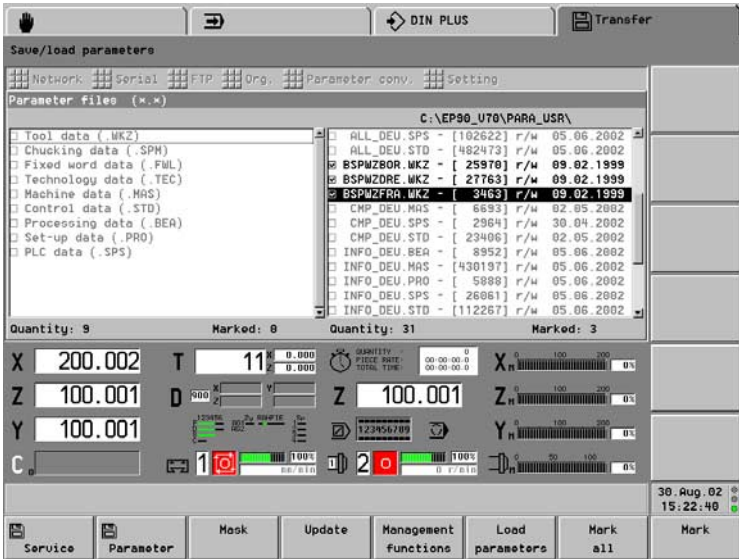
Mark all

Marks all displayed files or parameter/operating resources (selection) – pressing again unmarks them

Mark

Marks the selected file or the parameter/operating resource – pressing again unmarks it

- **Mouse operation:** You can use a mouse to position the cursor, and mark and open a file (with DIN PLUS programs, parameter files and operating resource files).



- During loading the CNC PILOT recognizes the parameter/operating resource group from the file-name extension. This is why the file name can be changed on external systems – but not the extension.
- During download the control checks whether the user is authorized to edit this parameter, or whether the Automatic operating mode is active. If the user is not authorized to edit the parameter, the parameter is skipped.

10.4.2 Saving Parameters and Operating Resources

Data backup of parameters and operating resources is conducted in two steps:

- ▶ Create backup files (backup function)
- ▶ Transfer backup files to an external system (standard transfer function)

Restoring a data backup of parameters and operating resources is also conducted in two steps:

- ▶ Retrieving backup files from the external system (Standard transfer function)
- ▶ Integrating backup files in CNC PILOT (Restore function)

Backup transfers the following files in the directory BACKUP:

- All parameter files
- All operating resource data
- All associated fixed-word lists
- Maintenance system files

Parameters and operating resource files are converted during backup.

Restore downloads all backup files into the directory BACKUP (except maintenance system files).

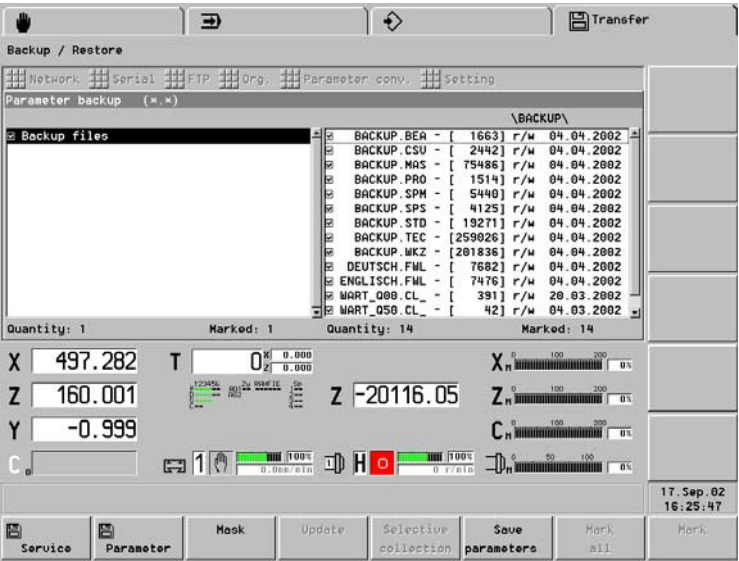
Call: Menu item “Parameter conversion—Backup / Restore”

Esc Back to Transfer menu

Operation

- Arrow up/down (only in right window); moves the cursor within the file list
- Horizontal arrow keys: move from left to right window and vice versa.
- Enter (only in right window): displays the file content—you close the file by pressing Enter again (or ESC key)

Mask	(Only in right window): Sort by data or file name
Save parameters	Make backup. All existing backup files are deleted. Then the new backup files are created.
Load parameters	Restore files.



Data backup soft keys

Mask	Define the sorting order
Update	Update the file list
Save parameters	Make backup
Load parameters	Restore files



Restore expects a **complete file group** saved by backup. HEIDENHAIN recommends that you always treat a file group saved by backup as a block.

Prerequisites for **Restore**:

- Log in as system manager
- The automatic mode must **not** be active
- The backup files must be available in the BACKUP directory

Maintenance-system files can be restored only by trained service personnel.

10.5 File Organization

The organization functions are used for CNC PILOT files and, under the following conditions, also for files of the communication partner:

- Windows network transfer method
- Login as system manager

Selection of the file organization:

- Organization menu item (only for local CNC PILOT files)
- Organizational functions soft keys.

Information in the file list

- File names and extensions (*.NC = main program; *.NCS = subprogram; etc.)
- File size in bytes – shown in brackets “[...]”
- Attribute
 - “r/w”: reading and writing allowed (read/write)
 - “ro”: only reading allowed (read only)
- Date and time of the last change

Switch the file group, change the “mask”

The current mask setting is displayed below the menu line.

Mask

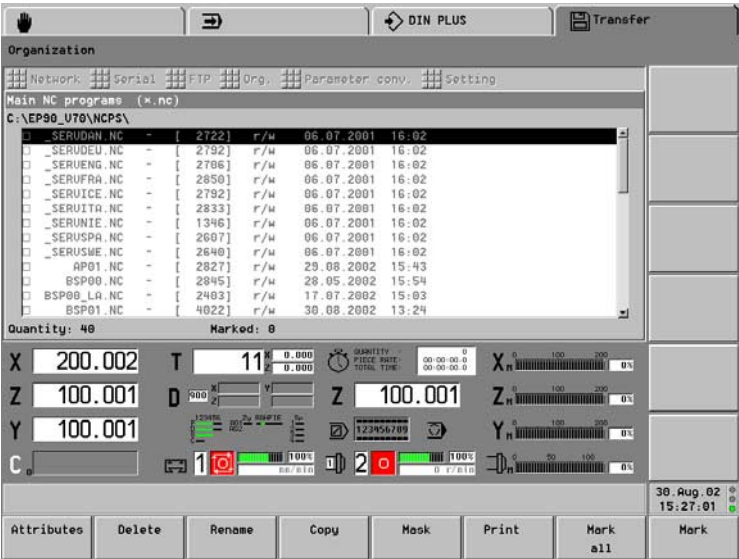
- **Data type:** See “10.3.1 Enabling, File Types”
- **Sorting:** Sort files sorted by name or date
- **Mask:** Only those entries are displayed that the mask allows.
Wildcards:
*: Represents any characters at this position.
?: Represents any one character at this position.
The CNC PILOT automatically adds “*” to any entered mask.

Operation:

- Vertical arrow keys; Page Up/Dn: move the cursor within the list of files.
- Enter character/character series: The cursor positions to the next file name beginning with this character sequence
- Enter (with DIN PLUS programs, parameters and operating resource files): Shows the file content. You close the file by pressing Enter again (or Esc).

Mark
all

Marks all displayed files – pressing again unmarks them



Organization-function soft keys

Delete	Delete marked files
Rename	Rename marked files
Copy	Copy marked files
Mask	Define the file type, sorting order, and masking
Print	Print marked files
Mark all	Mark all files
Mark	Mark file
Transfer functions	Call transfer functions
Management functions	Call organization functions

Continued ►

Mark

Or "+" (plus key) marks the selected file – pressing again unmarks the file

- **Mouse operation:** You can use a mouse to position the cursor, and mark and open a file (with DIN PLUS programs, parameter files and operating resource files).

For other **organization functions**, see the table of soft keys

01.NC	2843	
586.NC	2849	r/w
02.NC	410	r/w
03.NC	365	r/w
33.NC	557	r/w
4.NC	368	r/w
5.NC	368	r/w
6.NC	167	r/w
7.NC	3152	r/w
8.NC	1291	r/w
9.NC	1733	r/w
10.NC	11546	r/w
11.NC	13776	r/w
12.NC	2465	r/w
13.NC	2465	r/w
14.NC	2440	r/w
15.NC	2440	r/w



11

Tables and overviews

11.1 Undercut and Thread Parameters

11.1.1 Undercut DIN 76, Parameters

TURN PLUS determines the parameters for the thread undercut (undercut DIN 76) from the thread pitch.

Where:

Thr.pitch = thread pitch

I = depth of undercut (radius)

K = width of undercut

R = undercut radius

W = undercut angle

The undercut parameters are in accordance with DIN 13 for metric threads.

External thread				
Thr.pitch	I	K	R	W
0.2	0.3	0.7	0.1	30°
0.25	0,4	0.9	0,12	30°
0.3	0.5	1.05	0.16	30°
0.35	0.6	1.2	0.16	30°
0,4	0.7	1.4	0.2	30°
0.45	0.7	1.6	0.2	30°
0.5	0.8	1.75	0.2	30°
0.6	1	2.1	0,4	30°
0.7	1.1	2.45	0,4	30°
0.75	1.2	2.6	0,4	30°
0.8	1.3	2.8	0,4	30°
1	1.6	3.5	0.6	30°
1.25	2	4.4	0.6	30°
1.5	2.3	5.2	0.8	30°
1.75	2.6	6.1	1	30°
2	3	7	1	30°
2.5	3.6	8.7	1.2	30°
3	4.4	10.5	1.6	30°
3.5	5	12	1.6	30°
4	5.7	14	2	30°
4.5	6.4	16	2	30°
5	7	17.5	2.5	30°
55	7.7	19	3.2	30°
6	8.3	21	3.2	30°

Internal thread				
Thr.pitch	I	K	R	W
0.2	0.1	1.2	0.1	30°
0.25	0.1	1.4	0,12	30°
0.3	0.1	1.6	0.16	30°
0.35	0.2	1.9	0.16	30°
0,4	0.2	2.2	0.2	30°
0.45	0.2	2.4	0.2	30°
0.5	0.3	2.7	0.2	30°
0.6	0.3	3.3	0,4	30°
0.7	0.3	3.8	0,4	30°
0.75	0.3	4	0,4	30°
0.8	0.3	4.2	0,4	30°

Internal thread (continued)				
Thr.pitch	I	K	R	W
1	0.5	5.2	0.6	30°
1.25	0.5	6.7	0.6	30°
1.5	0.5	7.8	0.8	30°
1.75	0.5	9.1	1	30°
2	0.5	10.3	1	30°
2.5	0.5	13	1.2	30°
3	0.5	15.2	1.6	30°
3.5	0.5	17.7	1.6	30°
4	0.5	20	2	30°
4.5	0.5	23	2	30°
5	0.5	26	2.5	30°
55	0.5	28	3.2	30°
6	0.5	30	3.2	30°

11.1.2 Undercut DIN 509 E, Parameters

The undercut parameters are determined from the cylinder diameter;

Where:

- I = undercut depth
- K = undercut length
- R = undercut radius
- W = undercut angle

diameter	I	K	R	W
<= 1.6	0.1	0.5	0.1	15°
> 1.6 – 3	0.1	1	0.2	15°
> 3 – 10	0.2	2	0.2	15°
> 10 – 18	0.2	2	0.6	15°
> 18 – 80	0.3	2.5	0.6	15°
> 80	0,4	4	1	15°

11.1.3 Undercut DIN 509 F, Parameters

The undercut parameters are determined from the cylinder diameter;

Where:

- I = undercut depth
- K = undercut length
- R = undercut radius
- W = undercut angle
- P = transverse depth
- A = transverse angle

diameter	I	K	R	W	P	A
<= 1.6	0.1	0.5	0.1	15°	0.1	8°
> 1.6 – 3	0.1	1	0.2	15°	0.1	8°
> 3 – 10	0.2	2	0,4	15°	0.1	8°
> 10 – 18	0.2	2	0.6	15°	0.1	8°
> 18 – 80	0.3	2.5	0.6	15°	0.2	8°
> 80	0,4	4	1	15°	0.3	8°

11.1.4 Thread Parameters

To determine the thread parameters, the CNC PILOT uses the following table. Where $F = *$, the thread pitch—depending on the thread type—is determined from the diameter.

- Where:
- F = thread pitch
 - P = thread depth
 - R = thread width
 - A = thread angle, left
 - W = thread angle, right

The “backlash of threads” (ac) is determined from the thread pitch.

Thread pitch	ac
≤ 1	0.15
≤ 5	0.25
≤ 12	0.5
> 12	1

Calculation of Kb

$$Kb = 0.26384 \cdot F - 0.1 \cdot \sqrt{F}$$

Thread type Q		F	P	R	A	W
Q=1 Metric ISO fine-pitch thread (DIN 13)	Outside	–	$0.61343 \cdot F$	F	30°	30°
	Inside	–	$0.54127 \cdot F$	F	30°	30°
Q=2 Metric ISO thread (DIN 13)	Outside	*	$0.61343 \cdot F$	F	30°	30°
	Inside	*	$0.54127 \cdot F$	F	30°	30°
Q=3 Metric ISO tapered thread (DIN 158)	Outside	–	$0.61343 \cdot F$	F	30°	30°
Q=4 Metric ISO tapered thread (DIN 158)		–	$0.61343 \cdot F$	F	30°	30°
Q=5 Metric ISO trapezoid thread	Outside	–	$0.5 \cdot F + ac$	$0.633 \cdot F$	15°	15°
	Inside		$0.5 \cdot F + ac$	$0.633 \cdot F$	15°	15°
Q=6 Flat metr. trapezoid thread (DIN 308)	Outside	–	$0.3 \cdot F + ac$	$0.527 \cdot F$	15°	15°
	Inside		$0.3 \cdot F + ac$	$0.527 \cdot F$	15°	15°
Q=7 Metric buttress thread (DIN 513)	Outside	–	$0.86777 \cdot F$	$0.73616 \cdot F$	3°	30°
	Inside	–	$0.75 \cdot F$	F–Kb	30°	3°
Q=8 Cylindrical round thread (DIN 405)	Outside	*	$0.5 \cdot F$	F	15°	15°
	Inside	*	$0.5 \cdot F$	F	15°	15°
Q=8 Cylindrical Whitworth thread (DIN 11)	Outside	*	$0.64033 \cdot F$	F	27.5°	27.5°
	Inside	*	$0.64033 \cdot F$	F	27.5°	27.5°
Q=10 Tapered Whitworth thread (DIN 2999)	Outside	*	$0.640327 \cdot F$	F	27.5°	27.5°
Q=11 Whitworth pipe thread (DIN 259)	Outside	*	$0.640327 \cdot F$	F	27.5°	27.5°
	Inside	*	$0.640327 \cdot F$	F	27.5°	27.5°
Q=12 Nonstandard thread		–	–	–	–	–
Q=13 UNC US coarse thread	Outside	*	$0.61343 \cdot F$	F	30°	30°
	Inside	*	$0.54127 \cdot F$	F	30°	30°
Q=14 UNF US fine-pitch thread	Outside	*	$0.61343 \cdot F$	F	30°	30°
	Inside	*	$0.54127 \cdot F$	F	30°	30°

Continued ►

Thread type Q		F	P	R	A	W
Q=15 UNEF US extra-fine-pitch thread	Outside	*	0.61343*F	F	30°	30°
	Inside	*	0.54127*F	F	30°	30°
Q=16 NPT US taper pipe thread	Outside	*	0.8*F	F	30°	30°
	Inside	*	0.8*F	F	30°	30°
Q=17 NPTF US taper dryseal pipe thread	Outside	*	0.8*F	F	30°	30°
	Inside	*	0.8*F	F	30°	30°
Q=18 NPSC US cylindrical pipe thread with lubricant	Outside	*	0.8*F	F	30°	30°
	Inside	*	0.8*F	F	30°	30°
Q=18 NPFS US cylindrical pipe thread without lubricant	Outside	*	0.8*F	F	30°	30°
	Inside	*	0.8*F	F	30°	30°

11.1.5 Thread Pitch

Q=2 Metric ISO thread

Diameter	Thread pitch
1	0.25
1,1	0.25
1,2	0.25
1,4	0.3
1,6	0.35
1,8	0.35
2	0.4
2,2	0.45
2,5	0.45
3	0.5
3,5	0.6
4	0.7
4,5	0.75
5	0.8
6	1
7	1
8	1.25
9	1.25
10	1.5
11	1.5
12	1.75
14	2
16	2
18	2.5

Diameter	Thread pitch
20	2.5
22	2.5
24	3
27	3
30	3.5
33	3.5
36	4
39	4
42	4.5
45	4.5
48	5
52	5
56	5.5
60	5.5
64	6
68	6

Q=8 Cylindrical round thread

Diameter	Thread pitch
12	2.54
14	3.175
40	4.233
105	6.35
200	6.35

Q=9 CylindricalWhitworth thread

Thread	diameter	Thread
designation	(in mm)	pitch
1/4"	6.35	1.27
5/16"	7.938	1.411
3/8"	9.525	1.588
7/16"	11.113	1.814
1/2"	12.7	2.117
5/8"	15.876	2.309
3/4"	19.051	2.54
7/8"	22.226	2.822
1"	25.401	3.175
1 1/8"	28.576	3.629
1 1/4"	31.751	3.629
1 3/8"	34.926	4.233
1 1/2"	38.101	4.233
1 5/8"	41.277	5.08
1 3/4"	44.452	5.08
1 7/8"	47.627	5.645
2"	50.802	5.645
2 1/4"	57.152	6.35
2 1/2"	63.502	6.35
2 3/4"	69.853	7.257

Q=10TaperedWhitworth thread

Thread	diameter	Thread
designation	(in mm)	pitch
1/16"	7.723	0.907
1/8"	9.728	0.907
1/4"	13.157	1.337
3/8"	16.662	1.337
1/2"	20.995	1.814
3/4"	26.441	1.814
1"	33.249	2.309
1 1/4"	41.91	2.309
1 1/2"	47.803	2.309
2"	59.614	2.309
2 1/2"	75.184	2.309
3"	87.884	2.309
4"	113.03	2.309
5"	138.43	2.309
6"	163.83	2.309

Q=11 Whitworth pipe thread

Thread	diameter	Thread
designation	(in mm)	pitch
1/8"	9.728	0.907
1/4"	13.157	1.337
3/8"	16.662	1.337
1/2"	20.995	1.814
5/8"	22.911	1.814
3/4"	26.441	1.814
7/8"	30.201	1.814
1"	33.249	2.309
1 1/8"	37.897	2.309
1 1/4"	41.91	2.309
1 3/8"	44.323	2.309
1 1/2"	47.803	2.309
1 3/4"	53.746	2.309
2"	59.614	2.309
2 1/4"	65.71	2.309
2 1/2"	75.184	2.309
2 3/4"	81.534	2.309
3"	87.884	2.309
3 1/4"	93.98	2.309
3 1/2"	100.33	2.309
3 3/4"	106.68	2.309
4"	113.03	2.309
4 1/2"	125.73	2.309
5"	138.43	2.309
5 1/2"	151.13	2.309
6"	163.83	2.309

Q=13 UNC US coarse thread

Thread	diameter	Thread
designation	(in mm)	pitch
0.073"	1.8542	0.396875
0.086"	2.1844	0.453571428
0.099"	2.5146	0.529166666
0.112"	2.8448	0.635
0.125"	3.175	0.635
0.138"	3.5052	0.79375
0.164"	4.1656	0.79375
0.19"	4.826	1.058333333
0.216"	5.4864	1.058333333

Continued ►

Thread	diameter	Thread
designation	(in mm)	pitch
1/4"	6.35	1.27
5/16"	7.9375	1.411111111
3/8"	9.525	1.5875
7/16"	11.1125	1.814285714
1/2"	12.7	1.953846154
9/16"	14.2875	2.116666667
5/8"	15.875	2.309090909
3/4"	19.05	2.54
7/8"	22.225	2.822222222
1"	25.4	3.175
1 1/8"	28.575	3.628571429
1 1/4"	31.75	3.628571429
1 3/8"	34.925	4.233333333
1 1/2"	38.1	4.233333333
1 3/4"	44.45	5.08
2"	50.8	5.644444444
2 1/4"	57.15	5.644444444
2 1/2"	63.5	6.35
2 3/4"	69.85	6.35
3"	76.2	6.35
3 1/4"	82.55	6.35
3 1/2"	88.9	6.35
3 3/4"	95.25	6.35
4"	101.6	6.35

Q=14 UNF US fine-pitch thread

Thread	diameter	Thread
designation	(in mm)	pitch
0.06"	1.524	0.3175
0.073"	1.8542	0.352777777
0.086"	2.1844	0.396875
0.099"	2.5146	0.453571428
0.112"	2.8448	0.529166666
0.125"	3.175	0.577272727
0.138"	3.5052	0.635
0.164"	4.1656	0.705555555
0.19"	4.826	0.79375
0.216"	5.4864	0.907142857
1/4"	6.35	0.907142857
5/16"	7.9375	1.058333333
3/8"	9.525	1.058333333
7/16"	11.1125	1.27

Thread	diameter	Thread
designation	(in mm)	pitch
1/2"	12.7	1.27
9/16"	14.2875	1.411111111
5/8"	15.875	1.411111111
3/4"	19.05	1.5875
7/8"	22.225	1.814285714
1"	25.4	1.814285714
1 1/8"	28.575	2.116666667
1 1/4"	31.75	2.116666667
1 3/8"	34.925	2.116666667
1 1/2"	38.1	2.116666667

Q=15 UNEF US extra-fine-pitch thread

Thread	diameter	Thread
designation	(in mm)	pitch
0.216"	5.4864	0.79375
1/4"	6.35	0.79375
5/16"	7.9375	0.79375
3/8"	9.525	0.79375
7/16"	11.1125	0.907142857
1/2"	12.7	0.907142857
9/16"	14.2875	1.058333333
5/8"	15.875	1.058333333
11/16"	17.4625	1.058333333
3/4"	19.05	1.27
13/16"	20.6375	1.27
7/8"	22.225	1.27
15/16"	23.8125	1.27
1"	25.4	1.27
1 1/16"	26.9875	1.411111111
1 1/8"	28.575	1.411111111
1 3/16"	30.1625	1.411111111
1 1/4"	31.75	1.411111111
1 5/16"	33.3375	1.411111111
1 3/8"	34.925	1.411111111
1 7/16"	36.5125	1.411111111
1 1/2"	38.1	1.411111111
1 9/16"	39.6875	1.411111111
1 5/8"	41.275	1.411111111
1 11/16"	42.8625	1.411111111
1 3/4"	44.45	1.5875
2"	50.8	1.5875

Q=16 NPT US taper pipe thread

Thread	diameter	Thread
designation	(in mm)	pitch
1/16"	7.938	0.94074074
1/8"	10.287	0.94174074
1/4"	13.716	1.411111111
3/8"	17.145	1.411111111
1/2"	21.336	1.814285714
3/4"	26.67	1.814285714
1"	33.401	2.208695652
1 1/4"	42.164	2.208695652
1 1/2"	48.26	2.208695652
2"	60.325	2.208695652
2 1/2"	73.025	3.175
3"	88.9	3.175
3 1/2"	101.6	3.175
4"	114.3	3.175
5"	141.3	3.175
6"	168.275	3.175
8"	219.075	3.175
10"	273.05	3.175
12"	323.85	3.175
14"	355.6	3.175
16"	406.4	3.175
18"	457.2	3.175
20"	508.0	3.175
24"	609.6	3.175

Q=17 NPTF US taper dryseal pipe thread

Thread	diameter	Thread
designation	(in mm)	pitch
1/16"	7.938	0.94174074
1/8"	10.287	0.94174074
1/4"	13.716	1.411111111
3/8"	17.145	1.411111111
1/2"	21.336	1.814285714
3/4"	26.67	1.814285714
1"	33.401	2.208695652
1 1/4"	42.164	2.208695652
1 1/2"	48.26	2.208695652
2"	60.325	2.208695652
2 1/2"	73.025	3.175
3"	88.9	3.175

Q=18 NPSC US cylindrical pipe thread with lubricant

Thread	diameter	Thread
designation	(in mm)	pitch
1/8"	10.287	0.94174074
1/4"	13.716	1.411111111
3/8"	17.145	1.411111111
1/2"	21.336	1.814285714
3/4"	26.67	1.814285714
1"	33.401	2.208695652
1 1/4"	42.164	2.208695652
1 1/2"	48.26	2.208695652
2"	60.325	2.208695652
2 1/2"	73.025	3.175
3"	88.9	3.175
3 1/2"	101.6	3.175
4"	114.3	3.175

Q=19 NPFS US cylindrical pipe thread without lubricant

Thread	diameter	Thread
designation	(in mm)	pitch
1/16"	7.938	0.94174074
1/8"	10.287	0.94174074
1/4"	13.716	1.411111111
3/8"	17.145	1.411111111
1/2"	21.336	1.814285714
3/4"	26.67	1.814285714
1"	33.401	2.208695652

11.2 Technical Information

CNC PILOT 4290	
Standard features	Contouring control with integrated motor control and integrated inverter <ul style="list-style-type: none">■ 2 close-loop axes X1 and Z1 on slide 1■ 1 close-loop spindle
Expandable	To up to 10 closed loops <ul style="list-style-type: none">■ Maximum 6 slides■ Maximum 4 spindles■ Maximum 2 C axes
Display	15-inch TFT flat-panel display
Program memory	Hard disk
Interpolation	<div>Straight lines</div> <div>Circular arcs</div> <div>C axis</div> <div>Helices</div> <div>Look-ahead</div> <div>In 2 principal axes, optionally in 3 principal axes (maximum ±10 m)</div> <div>In 2 axes (maximum radius 100 m)</div> <div>Interpolation of linear axes X and Z with the C axis</div> <div>Superimposition of circular and linear paths</div> <div>Preliminary calculation of the contour speed profile under consideration of up to 20 blocks</div>
Feed rate	<ul style="list-style-type: none">■ Maximum input at 0.001 mm resolution: 400 m/min■ Input in mm/min or mm/revolution■ Constant cutting speed■ Feed rate with chip breaking
Data interfaces	<ul style="list-style-type: none">■ RS-232-C/V.24 with up to 38.4 kilobaud■ Ethernet 100 BaseT (maximum 100 megabaud)■ Printout over serial interface
Accessories	<div>CNC PILOT 4290</div> <div>PC software for programming and training for the CNC PILOT 4290 lathe control:</div> <div><ul style="list-style-type: none">■ Programming and Test Run■ Program management■ Operating resource data management■ Data backup■ Training</div>

Programming		CNC PILOT 4290
DIN editor	Programming in DIN 66025 (ISO 6983) format	
DIN PLUS	<ul style="list-style-type: none">■ Setup information on workpiece blank, material, tools, chucking equipment■ Expanded command set (IF...THEN...ELSE; WHILE...; SWITCH...CASE)■ Dialog-guided input and help graphics for every programming function■ Subprograms and variable programming■ Control graphics for workpiece blanks and finished parts■ Parallel programming■ Parallel simulation■ alphanumeric program name	
Cycles for contour description	<ul style="list-style-type: none">■ Standard workpiece blank forms■ Recesses■ Undercuts■ Thread■ Hole pattern for the front face and lateral surface (XY and ZY plane)■ Figure pattern for the front face and lateral surface (XY and ZY plane)	
Fixed cycles	<ul style="list-style-type: none">■ Area clearance longitudinal and transverse■ Recessing cycles radial and axial■ Recess turning cycles radial and axial■ Undercut cycles■ Parting cycles■ Thread cycles radial and axial (multiple threads, successive threads, taper threads, variable pitch)■ Drilling, deep-hole drilling, and tapping cycles (conventional/rigid) radial and axial (C axis and Y axis)■ Contour milling and pocket milling radial and axial (C axis and Y axis)■ Area milling, centric polygon milling radial and axial (Y axis)	
TURN PLUS – graphic programming (option)		
	Geometrical workpiece description for workpiece blank and finished part	
	Graphical geometry program for calculation and display of any length of a series of dimensioned or also nondimensioned contour points	
	<ul style="list-style-type: none">■ With simple input of standard form elements: Chamfer, rounding radii, undercuts, recesses, thread, fits■ With simple input of transformations: Shifting, rotating, mirroring, multiplying■ If more than one geometrical solution exists for calculated coordinates all of them are presented for selection.	

Programming	CNC PILOT 4290
C-axis machining	<ul style="list-style-type: none">■ Display and programming in 3 views (ZX, XC, ZC plane), and the unrolled lateral surface■ Hole and figure pattern in the XC and ZC plane■ Machining cycles for drilling and milling on front face and lateral surface.
Y-axis machining	<ul style="list-style-type: none">■ Display and programming in 3 views (ZX, XY, ZY plane), and the unrolled lateral surface■ Hole and figure pattern in the XY and ZY plane■ Machining cycles for drilling and milling in the XY and ZY planes
TURN PLUS – Graphical programming (option)	<p>Programming in individual work steps for the turning axis, C axis, Y axis and full-surface machining with:</p> <ul style="list-style-type: none">■ Calling tool and cutting data■ Individual selection and definition of the machining mode■ Direct graphic verification of simulated cutting and subsequent correction possibilities for rear side machining■ Rechucking with machine-specific expert program for rear-side machining■ Interactive generation of the work blocks for rechucking and the second setup
TURN PLUS – Automatic DIN PLUS program generation (option)	<p>Automatic NC programming for the turning axis, C axis, Y axis and full-surface machining</p> <ul style="list-style-type: none">■ Automatic tool selection■ Automatic turret assignment■ Automatic generation of the machining sequence in all machining planes■ Automatic cutting limit by chucking equipment■ Automatic rechucking with machine-specific expert program for rear-side machining■ Automatic generation of work blocks for rechucking and the second setup
Information system	<ul style="list-style-type: none">■ Information on the G functions■ Support for graphic programming in TURN PLUS■ Support for interactive programming in TURN PLUS■ Information on parameters and operating resource data■ Context-sensitive call of the Info system■ Topic search in table of contents and subject index

Programming CNC PILOT 4290	
Measuring (option)	
In the machine	For setting up and measuring workpieces in the "manual" and "automatic" modes using a touch probe
At external measuring stations	Downloading the results of measurement at an external measuring station for processing the measured data in the Automatic mode of operation: <ul style="list-style-type: none">■ Maximum of 16 measurement points■ Data interface: RS-232-C/V.24■ Data transfer protocol: 3964-R
Tool monitoring	
Tool life monitoring	Tool life monitoring according to time and piece number
Load monitoring	Breakage and wear monitoring through motor current evaluation <ul style="list-style-type: none">■ Maximum of 4 drives■ Depiction of load values through bar graphic or line graphic
Tool inspection	For checking the indexable inserts during machining; returning to the workpiece on the retraction path

11.3 Peripheral Interfaces

The CNC PILOT is fitted with the following connectors for connecting peripheral devices or PCs and for integrating the control into networks. For information on the connectors available on your lathe, refer to your machine manual.

Serial interface

Connector: 9-pin, D-sub, male

Pin	Signal RS-232	
2	TxD	Transmit Data
3	RxD	Receive Data
4	DTR	Data Terminal Ready
5	GND	Signal Ground
6	DSR	Data Set Ready
7	RTS	Request to Send
8	CTS	Clear to Send
Housing		External shield



The interface is linked to the external PC by direct electrical connection. This may lead to interference in the interface, resulting from different power-supply reference levels.

Measures:

- If possible, use the service jack on the machine for the PC.
- Engage/disengage the connection only when the machine and PC are switched off.
- The cable length must not exceed 20 m (66 ft). Use even shorter cables if there is strong electromagnetic interference.
- Recommendation: Use an adapter with electrical isolation.

Ethernet interface

Connector: RJ45 connector, female

Pin	Layout
1	TX+
2	TX–
3	REC+
4	Do not assign
5	Do not assign
6	REC–
7	Do not assign
8	Do not assign
Housing	External shield

Symbols

- # variable
 - In NC program interpretation 70
 - Input/Output 173
 - Programming 175
- \$ – Slide code
 - Editing 76
 - Execution 181
- / Skip level
 - Editing 76
 - Execution 181
- ? – Simplified geometry programming 65
- 3-D view 209
- 4-axis machining
 - Cycle G810 123
 - Cycle G820 125

A

- Absolute coordinates 7
- Acceleration (slope) G48 113
- Active tool 178
- Actual values in variables G901 170
- Additional axes 62
- Additive compensation
 - Compensation G149 120
 - Compensation G149 Geo 94
 - Display 53
 - Entry 45
- Address parameters
 - Fundamentals 64
 - Programming 65
- Angle cutter 372
- Angle offset
 - Angle offset, measuring during spindle synchronization G90 161
 - C-angle offset G905 161
- Angular data for C axis 62
- Arcs See Circular arc

- Area milling
 - IWG roughing/finishing 304
 - TURN PLUS machining attribute 267
- Assigning contour to operation 110
- Attributes
 - For overlay elements G39-Geo 93
 - For TURN PLUS contours 263
- Automatic mode 41
- Automatic working plan generation (AWG) 306
- Auxiliary axes 62
- Auxiliary commands for contour description 92
- Auxiliary contour
 - Entering the section code 75
 - In the simulation 197
 - Section code 83
- Auxiliary feed rate 396
- AWG 306
- Axis designations 7

B

- Bar (TURN PLUS) 228
- Basic block display
 - Automatic mode 48
 - Simulation 200
- Basic block mode
 - Automatic mode 42
 - Simulation 196
- Basic contour (TURN PLUS) 229
- BLANK (section code) 83
- Blank attributes (TURN PLUS) 263
- Block Display
 - Font size 48
 - Setting 48
- Block number
 - Fundamentals 63
 - Numbering 74
- Block number increment 73

B

Block references
 Contour display 72
 Fixed cycles 122
 Block, editing
 Exchanging blocks 77
 Inserting, copying, deleting 78
 Boring G72 144
 Button tool 371
 Buttons 15
 Bytes 19

C

C axis
 Angle data 7
 C-angle offset G905 161
 Configuration 62
 Contours machined with 67
 Fundamentals 3
 Reference diameter G120 148
 Selecting G119 148
 Standardizing G153 148
 Zero-point shift G152 148
 Calculator (TURN PLUS user aid) 269
 Cast part
 DIN PLUS cast part G21-Geo 84
 TURN PLUS workpiece blank 228
 Centering
 DIN PLUS cycle G72 144
 TURN PLUS
 Form element 238
 Front/rear face 244
 IWG machining 295
 Lateral surface 251
 Centering taper 394
 Centering tool 371
 Centric predrilling (IWG) 295
 Chains of tool dimensions G710 121

Chamfer
 DIN PLUS Cycle G88 139
 TURN PLUS form element 232
 Change – TURN PLUS contour 258
 Chuck part/cylinder tube G20-Geo 84
 Chucking equipment
 Displaying G65 159
 DIN PLUS section code 82
 Reference point 159
 Circular arcs
 DIN PLUS
 Front/rear face contour G102-, G103-Geo 97
 Front/rear face G102, G103 149
 Lateral surface contours G112/G113-Geo 103
 Lateral surface G112, G113 151
 Turning contour G2-, G3-, G12-, G13-Geo 85
 Turning cycles G2, G3, G12, G13 112
 TURN PLUS
 Basic contour 231
 Front face/rear side 243
 Lateral surface 250
 Circular interpolation 62
 Circular path. *See* Circular arc
 Circular pattern *see* Pattern
 Circular pattern with circular slots 108
 Circular saw blade 372
 Circular slot ccw
 DIN PLUS
 Front face G302-/G303-Geo 99
 Lateral surface G312-/G313-Geo 104
 In circular patterns 108
 TURN PLUS
 Front/rear face 247
 Lateral surface 254
 Comments
 Fundamentals 64
 Input in Edit menu 77
 Input in Geometry menu 75

C

Compensation

- Additive compensation G149 120
- Additive compensation G149-Geo 94
- Entering compensation values 44

Compensation of left/right tool point G151 121

Configuration

- DIN PLUS help graphic 74
- TURN PLUS 318

Connect (TURN PLUS contours) 262

Constant cutting speed Gx96 114

Constant feed (manual control) 26

Contour

- Activate/update contour display 74
- Contour display, switch-on 68
- Contour selection (simulation) 202
- Contour simulation 203
- Mirror/shift contour G121 117
- Section code in DIN PLUS 82

Contour – machining, arrangement 110

Contour definition

DIN PLUS

- Front face/rear side 96
- Fundamentals 66
- Geometry menu 75
- Lateral surface 102
- Main menu 73
- Workpiece blank/finished part contour 84

TURN PLUS

- Contour editing 256
- Contour elements, checking 270
- Contour of workpiece blank 228
- Entering basic contour 229
- Entering form elements 232
- Front face/rear side 242
- Fundamentals for workpiece description 219
- Lateral surface 249

Contour generation during simulation 67

Contour machining (finishing) IWG 298

Contour milling

DIN PLUS Cycle G840 152

TURN PLUS

- IWG machining 303
- Machining attribute 267

Contour of workpiece blank

DIN PLUS

- Blank, definition of 84
- Fundamentals 66

TURN PLUS

- Blank contour, editing 256
- Contour elements 228
- Input of 219

Contour recessing (IWG) 290

Contour regeneration

- Contour follow-up G703 164
- Contour follow-up, saving/loading G702 164
- Fundamentals 67
- In the simulation 206

Contour repeat cycle G83 136

Contour repeats (DIN PLUS example) 184

Contour-based turning cycles 122

Contourparallel roughing

DIN PLUS

- Cycle G830 126
- With neutral tool – cycle G835 127

TURN PLUS IWG machining 286

Contours for turning 66

Control graphics (TURN PLUS) 317

Control of the program run 183

Control parameters 344

Controlled parting

- By servo-lag monitoring G917 162
- By spindle monitoring G991 163
- Values for controlled parting G992 164

C

- Controls and Displays 13
 - Machine operating panel 13
 - Operating panel 13
 - Screen 13
 - Touch pad 13
- Conventional DIN programming 60
- Converting (parameters and operating resources) 416
- Converting and mirroring G30 169
- Coolant
 - Technology database 396
 - TURN PLUS 321
- Coordinates
 - Absolute 7
 - Coordinate system 7
 - Fundamentals 62
 - Incremental 8
 - Polar 8
 - Programming 65
- Coordinates, unknown 65
- Copy (TURN PLUS Contours) 226
- Copying tool 371
- Counterbore 371
- Counterboring (IWG) 295
- Countersinkers 371
- Countersinking
 - DIN PLUS cycle G72 144
 - TURN PLUS
 - Countersinking on front/rear face 245
 - Countersinking on lateral surface 252
 - Form element 238
 - IWG counterboring 295
 - IWG countersinking 295
- Countersinking (IWG) 295
- Cursor 19
- Cut display mode 203
- Cutter compensation, switching G148 120
- Cutting data (TURN PLUS IAG) 284

- Cutting limit
 - Fixing/editing (TURN PLUS) 277
 - With preparation (TURN PLUS) 273
 - With residual roughing (TURN PLUS) 287
- Cutting material
 - Designations, specifying 400
 - Technology database 395
- Cutting parameters
 - Finding in TURN PLUS 321
 - Technology database 395
- Cutting path display mode 197
- Cutting speed
 - Manual control 25
 - Technology database 396
- Cycle end G80 134
- Cycle specification (TURN PLUS IWG) 284

D

- D display 53
- Data Backup
 - General information 19
 - Transfer mode 408
- Data exchange (transfer) 408
- Data input 15
- Data input keyboard 2
- Data input/output (NC program) 173
- Data transfer
 - General 413
 - Installation of 410
 - Settings for FTP 411
 - Settings for Windows networks 410
 - Transfer directory 411
- Database for Chucking Equipment
 - Centering taper 394
 - Chuck 389
 - Chuck types, overview 388
 - Chucking equipment editor 386
 - Chucking equipment lists 387
 - Chucking equipment type 386

- Clamping jaws 390
- Collet 391
- Dead center 393
- Face driver 392
- General information 386
- ID number of chucking equipment 386
- Lathe center 393
- Mandrel 391
- Overview of chuck types 388
- Rotating gripper 392
- DataPilot 408
- Date, setting the 399
- Datum setting/canceling (simulation) 204
- Dead center 393
- Debug 210
- Deburring
 - DIN PLUS milling cycle G840 152
 - TURN PLUS machining attribute 268
- Deep-hole drilling G74 147
- Default value 19
- Deleting
 - TURN PLUS contour manipulation 259
 - TURN PLUS element input 226
- Deleting the chucking data 277
- Delta drill 371
- Diagnosis 404
- Dialog box 19
- Dialog texts with subprograms 182
- Digitizing (TURN PLUS user aid) 270
- Dimensions (simulation) 204
- DIN PLUS
 - Basic structure 60
 - Editor 71
 - Fundamentals 2
 - Main menu 72
 - Parallel editing 61
 - Programming 60
 - Screen 61
- Direction of contour description 66
- Direction of contour machining 66
- Directories, enabled 413
- Display of actual values 52
- Displays
 - Block display 48
 - DIN PLUS contour display 68
 - Machine display
 - Display fields, defining 348
 - Meaning of display elements 52
 - Switching in Automatic mode 52
 - Switching in Manual control 24
 - Simulation
 - Graphic elements 197
 - Note on the display modes 198
- Distance-to-go display 52
- Drilling
 - DIN PLUS
 - Bore hole (centric) G49-Geo 91
 - Cycle for boring, countersinking G72 144
 - Cycle for deep-hole drilling G74 147
 - Cycle for drilling G71 143
 - Cycle for tapping G36 146
 - Cycle for tapping G73 145
 - Front face/rear side G300-Geo 98
 - Fundamentals 66
 - Lateral surface G310-Geo 103
 - TURN PLUS
 - Centric bore hole 238
 - Hole on front face/rear side 244
 - Hole on lateral surface 251
 - IWG centric predrilling 295
 - IWG drilling 296
 - Machining attribute 266
- Drilling and slot milling tool 372
- Drilling tools 371
- Drop-down menu 14
- Dwell time G4 168

E

Eccentric polygon. *See* Polygon

Editing 19

Editing switch 399

Element dimensions (simulation) 204

Elements of the DIN program 63

Enabling

- Enabled directories 413
- Enabling name of the communications partner 411
- Enabling password (network) 405

Enabling overview (machine display) 53

End

- Pocket/island G309-Geo 96
- Section code 83

End mill 372

Engraving

- DIN PLUS Cycle G840 152
- TURN PLUS

 - IWG machining 305
 - Machining attribute 268

Equidistants 10

Error log file 405

Error message (simulation) 200

Error message 17

Esc key 15

Ethernet interface

- Connector assignment 433
- Transfer method with 409

Evaluate events 178

Examples

- Contour repetitions 184
- DIN PLUS Programming 184
- Full-surface machining with opposing spindle 187
- Full-surface machining with single spindle 192
- Programming Machining Cycles 184
- TURN PLUS 328

Expert programs 70

Exposed contours 66

Extended input for address parameters 66

Extension 19

External subprograms 70

F

F display 53

Face driver 392

Face roughing G820 124

Face turning, simple G82 135

Feed per minute

- Linear axes G94 114
- Manual control 25
- Rotary axes G192 113

Feed per revolution 25

Feed rate

- Constant G94 114
- Feed per minute, rotary axes G192 113
- Feed rate override, display 53
- Feed rate, interrupted G64 113
- Feed-rate override 100% G908 171
- Feed-rate override in automatic mode 44
- Feed-rate reduction G38-Geo 93
- In manual control mode 25
- Per revolution G95-Geo 94
- Per revolution Gx95 114
- Per tooth Gx93 114
- Rotary axes G192 113
- TURN PLUS attribute 263

File organization 419

File Transfer Protocol (FTP) 409

File types 413

Finished-part contour

- Fundamentals 66
- Section code FINISHED PART 83
- TURN PLUS 220

- Finishing tool 371
- Finish-machining
 - DIN PLUS
 - Cycle G890 132
 - Finishing feed rate 94
 - TURN PLUS IWG
 - Clearance turning 299
 - Contour machining (G890) 298
 - Hollowing (neutral tool) 301
 - Residual-contour machining 300
 - Undercuts 299
- Fits
 - IWG measuring step 299
 - TURN PLUS holes 324
- Fixed stop, traverse to .. G916 162
- Fixed-word list 400
- Forged part (TURN PLUS) 228
- Form elements
 - DIN PLUS 86
 - TURN PLUS 232
- Free editing
 - Fundamentals 72
 - Menu items 74
- Front face
 - Contour description 96
 - Fundamentals 62
 - Machining 149
 - Section code 83
- Front window (simulation) 201
- FTP (File Transfer Protocol) 409
- Full circle
 - DIN PLUS
 - Front/rear face G304-Geo 99
 - Lateral surface G314-Geo 105
 - TURN PLUS
 - Front/rear face 246
 - Lateral surface 253

- Full-surface machining
 - Fundamentals 4
 - in DIN PLUS 187
- TURN PLUS
 - AWG – machining information..... 324
 - AWG – machining sequence 307

G

- G commands, overview 3
- G functions
 - Manual turning operations 26
 - Selection from list of geometry functions 75
 - Selection from list of machining functions 76
- G functions for contour description
 - G0-Geo Starting point of contour 84
 - G100-Geo Starting point for front face 96
 - G101-Geo Line on front face 97
 - G102-Geo Arc on front face 97
 - G103-Geo Arc on front face 97
 - G10-Geo Surface roughness 92
 - G110-Geo Starting point on lateral surface 102
 - G111-Geo Line on lateral surface 102
 - G112-Geo Arc on lateral surface 103
 - G113-Geo Arc on lateral surface 103
 - G12-Geo Circular arc 85
 - G13-Geo Circular arc 85
 - G149-Geo Additive compensation 94
 - G1-Geo Line 85
 - G20-Geo Chuck part, cylinder/tube 84
 - G21-Geo Cast part 84
 - G22-Geo Recess (standard) 86
 - G23-Geo Recess (general) 86
 - G24-Geo Thread with undercut 87
 - G25-Geo Undercut contour 88
 - G2-Geo Circular arc 85
 - G300-Geo Hole on front face 98
 - G301-Geo Linear slot on front face 99
 - G302-Geo Circular slot on front face 99

- G**
- G303-Geo Circular slot on front face 99
 - G304-Geo Full circle on front face 99
 - G305-Geo Rectangle on front face 100
 - G307-Geo Polygon on front face 100
 - G308-Geo Start of pocket/island 95
 - G309-Geo End of pocket/island 96
 - G310-Geo Hole on lateral surface 103
 - G311-Geo Linear slot on lateral surface 104
 - G312—Geo Circular slot on lateral surface 104
 - G313-Geo Circular slot on lateral surface 104
 - G314-Geo Full circle on lateral surface 105
 - G315-Geo Rectangle on lateral surface 105
 - G317-Geo Polygon on lateral surface 105
 - G34-Geo Thread (standard) 90
 - G37-Geo Thread (general) 90
 - G38-Geo Feed rate reduction 93
 - G39-Geo Attributes for overlay elements 94
 - G3-Geo Circular arc 85
 - G401-Geo Linear pattern on front face 100
 - G402-Geo Circular pattern on front face 101
 - G411-Geo Linear pattern on lateral surface 106
 - G412-Geo Circular pattern on lateral surface 106
 - G49-Geo Bore hole (centric) 91
 - G7-Geo Precision stop on 92
 - G95-Geo Feed per revolution 94
 - G9-Geo Precision stop blockwise 92
 - G functions for machining
 - G0 Rapid traverse 110
 - G1 Linear movement 111
 - G100 Rapid traverse, front face/rear side 149
 - G101 Linear, front face/rear side 149
 - G102 Arc, front face/rear side 149
 - G103 Arc, front face/rear side 149
 - G110 Rapid traverse, lateral surface 150
 - G111 Linear, lateral surface 151
 - G112 Circular, lateral surface 151
 - G113 Circular, lateral surface 151
 - G119 Select C axis 148
 - G12 Circular movement 112
 - G120 Reference diameter 148
 - G121 Mirror and shift contour 117
 - G13 Circular movement 112
 - G14 Tool change position 110
 - G147 Safety clearance (milling cycles) 119
 - G148 Changing the cutter compensation 120
 - G149 Additive compensation 120
 - G15 Rotary axis traverse 168
 - G150 Compensate right tool tip 121
 - G151 Compensate left tool tip 121
 - G152 Zero point shift, C axis 148
 - G153 Standardize C axis 148
 - G162 Set synchronous mark 160
 - G192 per-minute feed, rotary axes 113
 - G2 Circular movement 112
 - G204 Waiting for time 170
 - G26 Spindle speed limit 113
 - G3 Circular movement 112
 - G30 Convert and mirror 169
 - G31 Thread cycle 140
 - G32 Simple thread cycle 141
 - G33 Thread, single path 142
 - G36 Tapping 146
 - G4 Dwell time 168
 - G40 Switch off TRC/MCRC 115
 - G41 Switch on TRC/MCRC 115
 - G42 Switch on TRC/MCRC 115
 - G47 Safety clearance 118
 - G48 Acceleration (slope) 113
 - G50 Switch off oversize 118
 - G51 Zero point displacement 116
 - G52 Switch off oversize 119
 - G53 Parameter-dependent zero-point shift 116
 - G54 Parameter-dependent zero-point shift 116
 - G55 Parameter-dependent zero-point shift 116
 - G56 Zero-point shift, additive 117
 - G57 Oversize, paraxial 119
 - G58 Oversize, contour-parallel 119

- G59 Zero shift, absolute 117
- G60 Switch off protection zone 169
- G62 One-sided synchronization 160
- G63 Synchronous start of slides 160
- G64 Interrupted feed rate 113
- G65 Chucking equipment 159
- G7 Precision stop on 168
- G701 Rapid traverse in machine coordinates 111
- G702 Save/load contour follow-up 164
- G703 Contour follow-up 164
- G706 K default branch 164
- G71 Drilling cycle 143
- G710 Add tool dimensions 121
- G717 Update nominal values 170
- G718 Ignore lag error 170
- G72 Boring, countersinking 144
- G720 Spindle synchronization 161
- G73 Tapping 145
- G74 Deep-hole drilling cycle 147
- G8 Precision stop off 168
- G80 Cycle end 134
- G81 Longitudinal turning, simple 134
- G810 Longitudinal roughing 122
- G82 Simple face roughing 135
- G820 Face roughing 124
- G83 Contour-repeat cycle 136
- G830 Contour-parallel roughing 126
- G835 Contour-parallel with neutral tool 127
- G840 Contour milling 152
- G845 Pocket milling, roughing 156
- G846 Pocket milling, finishing 157
- G85 Parting cycle 137
- G86 Simple recessing cycle 138
- G860 Contour-based recessing 128
- G866 Recessing cycle 129
- G869 Recess turning cycle 130
- G87 Line with radius 139
- G88 Line with chamfer 139
- G890 Contour finishing 132
- G9 Precision stop 168
- G901 Actual values to variables 170
- G902 Zero-point shift to variables 171
- G903 Servo lag to variables 171
- G905 C angle offset 161
- G906 Measuring angular offset during spindle synchronization 161
- G907 Block speed monitoring off 171
- G908 Feed rate override 100% 171
- G909 Interpreter stop 171
- G910 Switch on in-process measurement 165
- G912 Actual-value capture for in-process measuring 165
- G913 Switch off in-process measuring 165
- G914 Retract touch probe 165
- G915 Postprocess measurement 166
- G916 Traverse to a fixed stop 162
- G917 Controlled parting 162
- G918 Look-ahead 171
- G919 Spindle override 100% 171
- G920 Deactivate zero-point shifts 172
- G921 Deactivate zero-point shifts, tool lengths 172
- G93 Feed per tooth 114
- G94 Constant feed rate 114
- G95 Feed per revolution 114
- G96 Constant cutting speed 114
- G97 Spindle speed 114
- G975 Servo lag limit 172
- G98 Spindle with workpiece 169
- G980 Activate zero-point shifts 172
- G981 Activate zero-point shift, tool lengths 172
- G99 Workpiece group 110
- G991 Controlled parting – spindle monitoring 163
- G992 Values for controlled parting 164
- G995 Define monitoring zone 167
- G996 Type of load monitoring 167
- Geometry (in main menu) 73
- Geometry commands (DIN PLUS) 84
- Graphic (DIN PLUS) 74

Graphic display 49
 Graphic window 68
 Graphic, magnifying/reducing
 Simulation 208
 TURN PLUS 317
 Gripping device 372
 Guarding ring (TURN PLUS) 236

H

Handwheel 26
 Help 16
 Hollowing
 TURN PLUS IWG
 Cutting limit for 287
 Finishing (neutral tool) 301
 Finishing 300
 Hollowing – automatic 289
 Residual roughing, contour-parallel 288
 Residual roughing, longitudinal/transverse 287
 Roughing (neutral tool) 289
 TURN PLUS machining information 322

I

Identification number
 Chuck equipment 82
 Tool 80
 IF.. Program branch 180
 Illustrations for machine display 349
 Inches
 Defining the unit of measure 79
 Machine mode of operation 24, 41
 Programming 63
 Units of measure 8
 Incremental address parameters
 Code 64
 Programming 65
 Incremental coordinates 8
 Indexable-insert drills 371
 Infeed 396
 Info system 16

Information in variables 178
 Information on "unsolved geometric elements" 227
 In-process measuring
 Actual value capture for .. G912 165
 Switching off G913 165
 Switching on G910 165
 Touch probe, retracting G914 165
 INPUT (input of # variable) 173
 Input field 15
 Input resolution 429
 Input window 15
 INPUTA (input of V variable) 174
 Inputs/outputs
 Operator communication 64
 Programming 174
 Time for 70
 INS key 15
 Inserting (TURN PLUS contour) 260
 Inside machining (TURN PLUS machining information) 323
 Inspection mode 46
 Inspector (TURN PLUS user aid) 270
 Installation of data transfer 410
 Instructions, input 76
 Integer variable 175
 Interactive working plan generation (IWG) 282
 Interfaces
 Ethernet
 Connector assignment 433
 Transmission methods with 409
 Serial
 Configuration 412
 Connector assignment 433
 General information 409
 Intermediate contours 83
 Internal error 18
 Interpreter stop
 Interpreter stop G909 171
 Variable programming 179

- Interrupted feed rate G64 113
- Inverting (TURN PLUS contour) 262
- Island (DIN PLUS) 95
- Isolating a detail
 - Simulation 208
 - TURN PLUS 317
- IWG 282
- J**
- Jog keys 27
- K**
- Knurling tool 371
- L**
- L call 77
- Lag error (following error)
 - Ignoring G718 170
 - In variables G903 171
 - Lag error limit G975 172
- Language, selecting 399
- Lateral surface
 - Contour commands 102
 - Coordinate data 62
 - Machining commands 150
 - Reference diameter G120 148
 - Surface window (simulation) 201
 - TURN PLUS contours 249
- Lathe center 393
- Limit switch monitoring in simulation 207
- Line segment
 - DIN PLUS
 - Front/rear face contour G101-Geo 97
 - Front/rear face G101 149
 - Lateral surface contour G111-Geo 102
 - Lateral surface G111 151
 - Linear motion G1 111
 - Turning contour G1-Geo 85
 - With chamfer G88 139
 - With radius G87 139
 - TURN PLUS
 - Front/rear face 243
 - Lateral surface 250
 - Turning contour 230
- Linear and rotary axes 62
- Linear dimension 62
- Linear path. *See* Line segment
- Linear pattern *see* Pattern
- Linear slot
 - DIN PLUS
 - Front/rear face G301-Geo 99
 - Lateral surface G311-Geo 104
 - TURN PLUS
 - Front/rear face 247
 - Lateral surface 254
- Load display 53
- Load monitoring
 - Fundamentals 54
 - Limit values, editing 56
 - Load monitoring, type of G996 167
 - Monitoring zone, defining G995 167
 - Parameters for 58
 - Production under 55
 - Programming 167
 - Reference machining 54
 - Reference machining, analyzing 57
 - Working with the 57
- Load monitoring, type of G996 167
- Local subprograms 70
- Local variables 70
- Log file 405
- Longitudinal roughing G810 122
- Longitudinal turning, simple G81 134
- Look ahead G918 171

M

- M Commands
 - In manual control mode 25
 - Input 76
 - M00 program stop 183
 - M01 optional stop 183
 - M30 program end 183
 - M97 synchronous function 183
 - M99 program end with return jump 183
 - TURN PLUS IWG special machining 305
 - TURN PLUS program header 218
- Machine commands 183
- Machine data 25
- Machine dimensions, setting up 38
- Machine display
 - Adjusting/switching 52
 - Display elements 52
 - Display, defining 349
 - Fundamentals 12
- Machine operating panel 13
- Machine parameters 337
- Machine reference points 9
- Machine zero point 9
- Machining information (TURN PLUS) 320
- Machining modes (technology database) 395
- Machining modes TURN PLUS IWG
 - Drilling 295
 - Finishing 297
 - Milling 303
 - Recessing 290
 - Roughing 285
 - Thread 302
- Machining Parameters 353
- Machining sequence AWG
 - Editing 316
 - General information 307
 - List of 308
 - Managing 316
- Machining Simulation 205
- Machining with DIN PLUS
 - Machining commands 110
 - Machining menu 76
 - Section code 83
- Main cutting edge 69
- Main feed rate 396
- Maintenance system 401
- Manual control functions 24
- Manual direction keys 27
- Material (technology database) 395
- Material designations 400
- Mathematical expressions
 - Input in Edit menu 76
 - Input in Geometry menu 75
- Mathematical functions 175
- Measuring
 - In-process measuring 165
 - Postprocess measuring 166
 - TURN PLUS machining attribute 266
- Measuring optics 39
- Menu items 14
- Menu selection 19
- Metric
 - Dimensional system in automatic mode 41
 - Dimensional system in Manual control mode 24
 - Dimensional system, setting 79
 - Overview of units of measure 8
- Milling
 - DIN PLUS
 - Contour milling G840 152
 - Fundamentals 66
 - Pocket milling, finishing G846 157
 - Pocket milling, roughing G845 156
 - TURN PLUS
 - IWG milling 303
 - Machining attributes 267
- Milling contour position
 - DIN PLUS 95
 - TURN PLUS front/rear face 242

- TURN PLUS lateral surface 249
- Milling cutter radius compensation
 - Fundamentals 10
 - Programming 115
- Milling cycles
 - DIN PLUS
 - Contour milling G840 152
 - Pocket milling, finishing G846 157
 - Pocket milling, roughing G845 156
 - TURN PLUS
 - Area milling 304
 - Contour milling 303
 - Deburring 303
 - Engraving 305
- Milling depth
 - DIN PLUS 95
 - TURN PLUS – Front face/rear side 242
 - TURN PLUS – Lateral surface 249
- Milling direction (DIN PLUS)
 - Cycle G840 152
 - Cycle G845 156
 - Cycle G846 157
- Milling pins 372
- Milling tools 372
- Mirroring
 - DIN PLUS
 - Converting and mirroring G30 169
 - Mirror/shift contour G121 117
 - TURN PLUS
 - Auxiliary function 227
 - Manipulating contours 262
- Modal address parameters 65
- Modal G functions 65
- Modes of operation
 - Automatic mode 41
 - DIN PLUS 60
 - Manual control 24
 - Operating modes, selection 14
 - Overview 5

- Parameters 334
- Service and diagnosis 398
- Simulation 196
- Transfer 408
- TURN PLUS 216
- Motion Simulation 207
- Multiple tools
 - Definition of 380
 - Programming 69
- N**
 - Navigating 19
 - NC address parameters 64
 - NC blocks
 - Creating, deleting 71
 - Fundamentals 63
 - Numbering 73
 - NC commands
 - Editing, deleting 72
 - Fundamentals 63
 - NC drilling G72 144
 - NC program header 72
 - NC program management 72
 - NC program run 70
 - NC program run, checking 210
 - NC program sections 60
 - NC subprograms 70
 - Negative X coordinates 62
 - Nested contours 95
 - Networks
 - Installing 410
 - Overview 409
 - Settings (diagnosis) 405
 - New start (NC programs) 41
 - Nominal values, updating G717 170
 - Numeric keypad 14

O

- OK button 15
- One-sided synchronization G62 160
- Operating aids (TURN PLUS) 269
- Operating rights 398
- Operation
 - Buttons 15
 - Data input 15
 - Function selection 14
 - List operations 14
 - Menu bar 14
 - Soft-key row 14
- Operator communication 64
- Optional STOP
 - Automatic mode 44
 - M command M01 183
- Options 6
- Options 6
- Options, overview of 405
- Organization (file management) 419
- Outputs
 - # variable 173
 - Operator communication 64
 - Programming of 173
 - Time point of 70
 - V variables 174
- Overlay element (TURN PLUS)
 - Circular arc 239
 - Linear/circular overlay..... 240
 - Pontoon 240
 - Wedge 240
- Oversize
 - Blockwise G52-Geo 94
 - Contour parallel (equidistant) G58 119
 - Paraxial G57 119
 - Switch-off G50 118
 - TURN PLUS attribute 263

P

- Parallel editing (DIN PLUS) 65
- Parallel work 60
- Parameter
 - C-axis parameters 341
 - Control parameters 344
 - Editing 335
 - Linear axis parameters 342
 - Machine parameters 337
 - Machining parameters 353
 - Parameter groups 334
 - Protected parameters 336
 - Setup parameters 351
 - Spindle parameters 339
- Parameter description – subprograms 182
- Parameter values, reading (DIN PLUS) 175
- Parameter-dependent zero shift
 - G53..G55 116
- Parameters/operating resources
 - Converting 416
 - Saving 418
 - Transferring 416
- Parting (IWG)
 - Standard machining 292
- Parting tool 371
- Password 398
- Path 411
- Patterns
 - DIN PLUS
 - Circular front/rear face G402-Geo 101
 - Circular lateral side G412-Geo 106
 - Linear front/rear face G401-Geo 100
 - Linear lateral surface G411-Geo 106
- TURN PLUS
 - Circular front/rear face 248
 - Circular lateral surface 255
 - Linear front/rear face 248
 - Linear lateral surface 254
- Peak-to-valley height

- DIN PLUS command G10-Geo 92
 - Machining parameters 353
 - TURN PLUS attribute 263
 - Peripheral interfaces 433
 - PLC message 18
 - Pocket milling
 - Finishing G846 157
 - Milling contour, pocket 95
 - Roughing G845 156
 - Point dimensions (simulation) 204
 - Polar coordinates 8
 - Polygon
 - DIN PLUS
 - Front/rear face G307-Geo 100
 - Lateral surface G317-Geo 105
 - TURN PLUS
 - Front/rear face 247
 - Lateral surface 253
 - Position display 52
 - Position nominal values, updating G717 170
 - Post-process measuring
 - Cycle G915 166
 - Status 51
 - Precision stop
 - Blockwise G9 168
 - Blockwise G9-Geo 92
 - Off G8 168
 - Off G8-Geo 92
 - On G7 168
 - On G7-Geo 92
 - TURN PLUS Attribute 264
 - Predrilling (IWG) 295
 - Prepare (TURN PLUS) 273
 - Principal axes
 - Arrangement 7
 - Fundamentals 62
 - PRINT (output # variables) 173
 - PRINTA (output V variables) 174
 - Printer 409
 - Program blocks, moving 78
 - Program branch
 - Fundamentals 64
 - Programming 179
 - Program branch IF. 180
 - Program branching, SWITCH.. 181
 - Program compilation 70
 - Program end with return jump M99 183
 - Program execution 70
 - Program Head
 - DIN PLUS 79
 - TURN PLUS 218
 - Program memory 429
 - Program number 63
 - Program repetition, WHILE.. 180
 - Program run modification 43
 - Program section codes 79
 - Program selection 41
 - Program stop M00 183
 - Programming Machining Cycles
 - Notes on programming 69
 - Programming example 184
 - Protection zone
 - Defining 36
 - Protection zone monitoring (simulation) 205
 - Switching off G60 169
- Q**
- Quantity information 53
 - Quantity, monitoring for number of parts produced
 - Quantity default 43
 - Quantity in variable 178
 - Quantity information 53
- R**
- Radius G87 139
 - Rapid paths (simulation) 197
 - Rapid traverse
 - Front face/rear side G100 149
 - In machine coordinates G701 111

R

- Rapid traverse
 - Lateral surface G110 150
 - Rapid traverse G0 110
- Real variables 175
- Reamer 371
- Reaming
 - Cycle G72 144
 - IWG machining 295
- Rear-face machining
 - DIN PLUS
 - Elements of the front/rear face contour 96
 - Example with opposing spindle 187
 - Example with single spindle 192
 - Section code 83
 - Section code, programming 75
 - TURN PLUS
 - Machining information 324
 - Machining sequence 307
- Recess turning
 - DIN PLUS cycle G869 130
 - IWG machining 291
- Recess turning tool 371
- Recessing
 - DIN PLUS
 - Contour based recessing G860 128
 - Recess contour (general) G23-Geo 86
 - Recess contour (standard) G22-Geo 86
 - Recessing cycle G866 129
 - Simple G86 138
 - Simple G866 129
 - TURN PLUS
 - Form element, general recess 235
 - Form element, recess type D (sealing ring) 235
 - Form element, recess type F (relief turn) 236
 - Form element, recess type S (sealing ring) 236
 - IWG machining 290
- Recessing
 - DIN PLUS
 - Recess cycle G866 129
 - Recessing G860 128
 - TURN PLUS
 - IWG contour recessing 290
 - IWG recessing 290
- Recessing tool 371
- Rechucking 277
- Rectangle
 - DIN PLUS
 - Front/rear side G305-Geo 100
 - Lateral surface G315-Geo 105
 - TURN PLUS
 - Front/rear side 246
 - Lateral surface 253
- Reference diameter
 - Reference diameter G120 148
 - Section code 75
- Reference marks, traversing 22
- Reference plan, setting (TURN PLUS) 224
- Reference plane
 - Reference plane G308 95
 - Section code 75
- Reference point 9
- Relief turn
 - Form element G23-Geo 86
 - TURN PLUS form element 236
- Remote diagnosis 405
- Repetition factor of subprograms 70
- Replacement tool 69
- Residual-contour machining
 - DIN PLUS residual finishing 132
 - TURN PLUS
 - IWG contour parallel roughing 288
 - IWG cutting limitation 287
 - IWG finishing 300
 - IWG roughing 287
- Resolution (TURN PLUS) 262

- Restart 41
 - RETURN (section code) 83
 - Rotary axis
 - Feed per minute, rotary axes G192 113
 - Fundamentals 62
 - Moving G15 168
 - Rotating gripper 392
 - Roughing
 - DIN PLUS
 - Contour-parallel roughing G830 126
 - Contour-parallel with neutral tool G835 127
 - Face roughing G820 124
 - Longitudinal roughing G810 122
 - TURN PLUS
 - Automatic 286
 - Contour parallel 286
 - Hollowing with neutral tool 289
 - Longitudinal, transverse 285
 - Roughing tool 371
 - Rounding
 - DIN PLUS cycle G87 139
 - TURN PLUS form element 232
 - Run-out length (thread) 140
- S**
- Safety clearance
 - Turning G47 118
 - Milling G147 119
 - Screen display, configuring 74
 - Screen displays
 - DIN PLUS screen 61
 - General information 12
 - Simulations screen 196
 - Sealing ring (TURN PLUS form element) 235
 - Search functions 73
 - Secondary machining direction (NBR) 381
 - Section code in DIN PLUS
 - Entry in geometry menu 75
 - Entry in main menu 73
 - Overview 79
 - Semiautomatic (IWG) 282
 - Sending/receiving files 414
 - Separation point
 - TURN PLUS attribute 264
 - TURN PLUS machining information 326
 - Sequential events 178
 - Serial interface
 - Configuration 412
 - Connector assignment 433
 - General information 409
 - Service functions 398
 - Setting up the chucking table 37
 - Setup
 - DIN PLUS program header 79
 - Setup functions 34
 - Setup parameters 351
 - TURN PLUS program header 218
 - Shaft machining (TURN PLUS)
 - Machining information 326
 - Preparing a machining process 273
 - Shift (TURN PLUS contour) 261
 - Shifting the contour G121 117
 - Side milling cutter 372
 - Side view (YZ) (simulation) 201
 - Simple tools
 - Programming 81
 - Setup 28
 - Simulation
 - 3-D view 209
 - Chucking equipment display 197
 - Contour generation during simulation 205
 - Contour simulation 203
 - Dimensions 204
 - Displays 198
 - Errors and warnings 200
 - Front window 201
 - Graphic elements 197
 - Lines and path display 197
 - Machining Simulation 205

- S**
- Simulation
 - Main menu 201
 - Motion Simulation 207
 - NC program run, checking 210
 - Operating mode 196
 - Protection zone and limit switch monitoring 205
 - Screen contents 196
 - Side view (YZ) 201
 - Surface window 201
 - Synchronous point analysis 213
 - Time calculation 212
 - Tool representation 197
 - TURN PLUS control graphics 317
 - Zoom 208
 - Single hole (TURN PLUS) 244
 - Single-block mode
 - Automatic mode 43
 - Simulation 196
 - Skip cycle 181
 - Skip level:
 - Editing 76
 - Entering 43
 - Execution 181
 - Fundamentals 64
 - Slide change key 27
 - Slide code
 - Conditional block run 181
 - Fundamentals 64
 - Programming 76
 - Slide display 53
 - Slide synchronization 160
 - General information 160
 - One-sided synchronization G62 160
 - Synchronizing mark, setting G162 160
 - Synchronous start of slides G63 160
 - Slots
 - DIN PLUS
 - Circular slot on front/rear face G302-/G303-Geo 99
 - Circular slot on lateral surface G312-/G313-Geo 104
 - Linear slot on front/rear face G301-Geo 99
 - Linear slot on lateral surface G311-Geo 104
 - TURN PLUS
 - Circular slot on front/rear side 247
 - Circular slot on lateral surface 254
 - Linear slot on front/rear side 247
 - Linear slot on lateral surface 254
 - Soft-key row 14
 - Software handshake (data transmission) 412
 - Software limit switch
 - Manual control 24
 - Reference run 22
 - Source block display – simulation 202
 - Special machining (IWG) 305
 - Specifications 429
 - Spindle
 - Spindle change key 27
 - Spindle display 53
 - Spindle keys 27
 - Spindle override 100% G919 171
 - Spindle speed 25
 - Spindle status 53
 - Spindle synchronization G720 161
 - With workpiece G98 169
 - Spindle point stop 25
 - Spindle speed
 - Constant cutting speed Gx96 114
 - Speed limitation Gx26 113
 - Speed monitoring blockwise off G907 171
 - Spindle speed Gx97 114
 - Spindle speed override 44
 - Start block search 42
 - Starting a pocket/island G308 Geo 95
 - Starting length (thread) 140
 - Starting point of contour
 - DIN PLUS
 - Displaying 68

Front/rear face G100-Geo 96
 Lateral surface G110-Geo 102
 Turning contour G0-Geo 84
 TURN PLUS
 Basic contour 229
 Front/rear face 242
 Lateral surface 249
 Step drill 371
 Stopper tool 372
 Structured DIN PLUS program 60
 Subroutine
 Call 182
 Fundamentals 70
 Section code 83
 SWITCH..CASE – Program branch 181
 Switch-off 23
 Switch-on 22
 Synchronization
 Synchronization, spindle G720 161
 Synchronizing function M97 183
 Synchronizing mark, setting G162 160
 Synchronous start of slides G63 160
 Synchronous point analysis 213
 System error 18

T

T commands
 Fundamentals 68
 Tool changing 120
 T display 52
 T number 80
 Tables
 Thread parameters 424
 Thread pitch 425
 Q= 2 Metric ISO thread 425
 Q= 8 Cylindrical round thread 425
 Q= 9 Cylindrical Whitworth thread 426
 Q=10 Taper Whitworth thread 426
 Q=11 Whitworth pipe thread 426

 Q=13 UNC US coarse thread 426
 Q=14 UNF US fine thread 427
 Q=15 UNEF US extrafine thread 427
 Undercut parameters DIN 509 E 423
 Undercut parameters DIN 509 F 423
 Undercut parameters DIN 76 422
 Tap 371
 Tap drill 371
 Tapping
 DIN PLUS
 Cycle G36 146
 Thread, contour-based G73 145
 TURN PLUS
 Centric bore hole 238
 Front face/rear side 246
 IWG machining 295
 Lateral surface 252
 Technology database
 Auxiliary feed rate 396
 Coolant 396
 Cutting material 395
 Cutting speed 396
 Infeed 396
 Machining operation 395
 Main feed rate 396
 Material 395
 Template control 70
 Thread cutter 372
 Thread overrun 140
 Thread parameters 424
 Thread pitch 425
 Thread tool, standard 371
 Thread undercut 137
 Threads
 DIN PLUS
 General G37-Geo 90
 Single path G33 142
 Standard G34-Geo 90
 Tapping G36 146

T

Threads

DIN PLUS

Thread cycle G31 140

Thread cycle, simple G32 141

With undercut G24-Geo 87

TURN PLUS

Form element 237

IWG machining 302

Machining attributes 265

Tilt position of tool carrier 68

Time calculation 212

Time of day, setting 399

Tool

Changing (DIN PLUS) 120

Measuring 39

Tool display (simulation) 197

Tool graphic, displaying 370

Tool call (TURN PLUS IWG) 283

Tool change point

Approach G14 110

Set 34

Tool compensation

Finding 40

Fundamentals 10

In automatic mode 44

Variable programming 178

Tool database

Adapter 385

Compensation values 381

Cutting length 381

Cutting speed (CSP) compensation 382

Deep compensation 382

Direction of rotation 381

Execution 381

Extended input 81

Feed-rate (FDR) compensation 382

Fixed-word list 381

General information 368

Mount type 382

Mounting position 385

Multipoint tools 380

NBR (secondary machining direction) 381

Notes on tool data 381

Overhang length 382

Picture number 381

Position angle 382

Setting dimensions 381

Simple tool 81

Tool editor 368

Tool graphic, displaying 370

Tool holder 383

Tool ID number 368

Tool life monitoring 380

Tool lists 369

Tool position 370

Width "dn" 381

Tool dimensions, adding G710 121

Tool edge compensation G148 120

Tool edge number 69

Tool graphic, displaying 370

Tool interchange chain

Defining exchange tools 33

Fundamentals 69

Tool length 10

Tool life management

Data display 28

Data in the tool database 380

In automatic mode 45

Parameter entry 33

Tool diagnosis bits 178

Tool life monitoring

Diagnosis bits 178

Fundamentals 69

Parameter entry 33

With load monitoring 167

- Tool list
 - Compare with NC program 31
 - Setup (machine setup) 29
 - Setup (TURN PLUS) 280
 - Transferring from NC program 32
- Tool movement without machining 110
- Tool programming 68
- Tool radius compensation
 - Fundamentals 10
 - Programming 115
- Tool selection
 - Manual control 25
 - TURN PLUS 320
- Tool types
 - Angle cutter 372
 - Bar grippers 372
 - Button tool 371
 - Centering tool 371
 - Circular saw blade 372
 - Copying tool 371
 - Counterbore 371
 - Countersinkers 371
 - Delta drill 371
 - Drilling and slot milling tool 372
 - Drilling tools 371
 - End mill 372
 - Finishing tool 371
 - Gripping device 372
 - Indexable-insert drills 371
 - Knurling tool 371
 - Milling pins 372
 - Milling tools 372
 - NC center tool 371
 - Parting tool 371
 - Reamer 371
 - Recess turning tool 371
 - Recessing tool 371
 - Roughing tool 371
 - Side milling cutter 372
 - Special drilling tool 375
 - Special milling cutter 372
 - Special turning tool 371
 - Step drill 371
 - Stopper tool 372
 - Tap 371
 - Tap drill 371
 - Thread cutter 372
 - Thread tool, standard 371
 - Touch probe 372
 - Turning tools 371
 - Turn-out tool 371
 - Twist drill 371
 - Workpiece handling systems 372
- Tools with more than one cutting edge 69
- Touch pad 13
- Touch probe
 - In-process measuring with 165
 - Tool 372
 - Tools, measuring with 39
- Transfer 408
- Transfer methods 409
- Transfer values for subprograms 182
- Transformations (TURN PLUS contours) 261
- Translation of the NC program 70
- Trimming (TURN PLUS contour) 256
- Tube (TURN PLUS) 228
- TURN PLUS
 - AWG
 - Editing and managing machining sequences 316
 - List of machining sequences 308
 - Machining sequence 307
 - Working plan generation 306
 - Contour definition
 - Assigning attributes 263
 - Blank contour, editing 256
 - Colors for selection points 225
 - Connect 262
 - Contour train, integrating 222

T

TURN PLUS

- Contour definition
 - Contour, deleting 259
 - Contour, editing 258
 - Contour, trimming 256
- Contours of the lateral surface 249
- Element entry, auxiliary functions 226
- Elements for C-axis contours 242
- Elements for finished part contours 229
- Entering the C-axis contours 223
- Entering the finished part contour 220
- Entering the workpiece blank contour 219
- Form elements 232
- Form elements, superimposing 221
- Inserting in the contour 260
- Machining attributes 265
- Notes on operation 225
- Overlay elements 239
- Selection with soft keys 225
- Selection with the touch pad 225
- Solving (form elements, figures, patterns) 262
- Transformations 261
- User aids 269
- Workpiece blank, contours 228
- Workpiece blanks, attributes 263
- Workpiece description 219
- General information
 - Configuration 318
 - Control graphic 317
 - Example 328
 - Machining information 320
 - Managing files 217
 - Operating mode 216
 - Program header 218
 - User aid 216

IWG

- Cutting data 284
- Cycle specification 284
- Drilling 295
- Finishing 297
- Interactive working plan generation 282
- Milling 303
- Recessing 290
- Roughing 285
- Special machining (SM) 305
- Thread cutting 302
- Tool call 283
- Machining information
 - Coolant 321
 - Cutting parameters 321
 - Drilling 324
 - Full-surface machining 324
 - Hollowing 322
 - Inside contour 322
 - Shaft machining 326
 - Tool selection 320
 - Turret assignment 320
- Preparing a Machining Process
 - Cutting limitation, defining 277
 - Rechucking 277
 - Tool list, setting up 280
 - Workpiece, clamping 273
- Turning (TURN PLUS contour) 261
- Turning contours 66
- Turning cycles
 - Contour-based 122
 - Simple 134
- Turning tools 371
- Turn-out tool 371
- Turret
 - DIN PLUS section code 80
 - DIN PLUS tool programming 68
 - TURN PLUS turret assignment 320
- Twist drill 371

U

Undercut

DIN PLUS

- Cycle G85 137
- Definition with G25-Geo 88
- DIN 509 E 88
- DIN 509 F 89
- DIN 76 89
- Form H 89
- Form K 90
- Form U 88

TURN PLUS

- DIN 509 E 232
- DIN 509 F 233
- DIN 76 233
- Form H 233
- Form K 234
- Form U 234

Undercut parameters

- DIN 509 E 423
- DIN 509 F 423
- DIN 76 422

Units of measurement

- In the DIN PLUS program 63
- Overview 8
- Unit of measure, setting 79

User, entering 398

V

Values for controlled parting G992 164

Variables

- # variables 175
- As address parameters 66
- Assignment 179
- Calculations 175
- Information in variables 178
- Input in Geometry menu 75
- Input/output of # variables 173
- Input/output of V variables 174

Machining menu, input 76

Programming 175

V variables 177

Validity range 177

Variable display 80

Verification of NC program run 210

VGP – simplified geometry programming 65

Views of contour..... 203

W

Wait for moment G204 170

Warnings (simulation) 200

WHILE.. Program repetition 180

WINDOW (special output window) 173

Window selection

DIN PLUS contour display 74

Simulation 201

WINDOWA (special output window) 174

WINDOWS networks 409

Working plan generation TURN PLUS

AWG 306

IWG 282

Working plane 67

Working window 12

Workpiece group G99 110

Workpiece handling systems 372

Workpiece transfer 161

Angle offset, measuring during spindle
synchronization G90 161

C-angle offset G905 161

Controlled parting using
spindle monitoring G991 163Controlled parting using lag
error (following error) 162

fixed stop, traversing to G916 162

Spindle synchronization G720 161

Values for controlled parting G992 164

W

Workpiece zero point

Entering 35

Fundamentals 9

Parameters 337

Workpiece, chucking (TURN PLUS) 273

Y

Y axis 3

Y-axis machining 67

Z

Zero point

C axis 62

Changing in TURN PLUS 226

Machine zero point 9

Shift absolute G59 117

Shift in the simulation 199

Shift in variables G902 171

Shift, activating G980 172

Shift, activating tool lengths G981 172

Shift, additive G56 117

Shift, C axis G152 148

Shift, deactivating G920 172

Shift, deactivating tool lengths G921 172

Shift, parameter-dependent G53..G55 116

Shift, relative G51 116

Shifts, overview 116

Workpiece zero 9

Zoom function

Automatic mode (graphical display) 49

Simulation 208

TURN PLUS control graphics 317

Connection between geometry and machining commands

Turning			
Function	Geometry	Cycle	
Individual elements	G0..G3 G12/G13	G810	Longitudinal roughing cycle
		G820	Face roughing cycle
		G830	Contour-parallel roughing cycle
		G835	Contour-parallel with neutral tool
		G860	Universal recessing cycle
		G869	Recess turning cycle
		G890	Finishing cycle
Recesses	G22 (standard)	G860	Universal recessing cycle
		G866	Simple recessing cycle
		G869	Recess turning cycle
Recesses	G23	G860	Universal recessing cycle
		G869	Recess turning cycle
Threads with undercut	G24	G810	Longitudinal roughing cycle
		G820	Face roughing cycle
		G830	Contour-parallel roughing cycle
		G890	Finishing cycle
		G31	Thread cycle
Undercut	G25	G810	Longitudinal roughing cycle
		G890	Finishing cycle
Threads	G34 (standard)	G31	Thread cycle
	G37 (general)		
Bore holes	G49 (turning center)	G71	Simple drilling cycle
		G72	Boring, countersinking, etc.
		G73	Tapping cycle
		G74	Deep-hole drilling cycle

C-axis machining – Front/Rear Face

Function	Geometry		Cycle	
Individual elements	G100..G103		G840 G845/G846	Contour milling Pocket milling, roughing/finishing
Figures	G301 G302/G303 G304 G305 G307	Linear slot Circular slot ccw Full circle Rectangle Eccentric polygon	G840 G845/G846	Contour milling Pocket milling, roughing/finishing
Bore holes	G300		G71 G72 G73 G74	Simple drilling cycle Boring, countersinking, etc. Tapping cycle Deep-hole drilling cycle

C-axis machining – lateral surface

Function	Geometry		Cycle	
Individual elements	G110..G113		G840 G845/G846	Contour milling Pocket milling, roughing/finishing
Figures	G311 G312/G313 G314 G315 G317	Linear slot Circular slot ccw Full circle Rectangle Eccentric polygon	G840 G845/G846	Contour milling Pocket milling, roughing/finishing
Bore holes	G310		G71 G72 G73 G74	Simple drilling cycle Boring, countersinking, etc. Tapping cycle Deep-hole drilling cycle

Overview of G commands for contour definition

Turning

Definition of workpiece blank		Page
G20-Geo	Chuck part, cylinder/tube	84
G21-Geo	Cast part	84
Basic contour elements		Page
G0-Geo	Starting point of contour	84
G1-Geo	Line segment	85
G2-Geo	Circular arc with incr. center dimensioning	85
G3-Geo	Circular arc with incr. center dimensioning	85
G12-Geo	Circular arc with abs. center dimensioning	85
G13-Geo	Circular arc with abs. center dimensioning	85
Contour form elements		Page
G22-Geo	Recess (standard)	86
G23-Geo	Recess/relief turn	86
G24-Geo	Thread with undercut	87
G25-Geo	Undercut contour	88
G34-Geo	Thread (standard)	90
G37-Geo	Thread (general)	90
G49-Geo	Bore hole at turning center	91
Help commands for contour definition		Page
Overview:	Help commands for contour definition	92
G7-Geo	Precision stop ON	92
G8-Geo	Precision stop OFF	92
G9-Geo	Precision stop blockwise	92
G10-Geo	Peak-to-valley height	92
G38-Geo	Feed rate reduction factor	93
G39-Geo	Attributes of superimposed elements	93
G52-Geo	Blockwise oversize	94
G95-Geo	Feed per revolution	94
G149-Geo	Additive correction	94

C-axis machining

Superimposed contours		Page
G308-Geo	Beginning of pocket/island	95
G309-Geo	End of pocket/island	96
End face contours		Page
G100-Geo	Starting point of end face contour	96
G101-Geo	Line segment on end face	97
G102-Geo	Circular arc on end face	97
G103-Geo	Circular arc on end face	97
G300-Geo	Bore hole on end face	98
G301-Geo	Linear slot on end face	99
G302-Geo	Circular slot on end face	99
G303-Geo	Circular slot on end face	99
G304-Geo	Full circle on end face	99
G305-Geo	Rectangle on end face	100
G307-Geo	Eccentric polygon on end face	100
G401-Geo	Linear pattern on end face	100
G402-Geo	Circular pattern on end face	101
Lateral surface contours		Page
G110-Geo	Starting point of lateral surface contour	102
G111-Geo	Line segment on lateral surface	102
G112-Geo	Circular arc on lateral surface	103
G113-Geo	Circular arc on lateral surface	103
G310-Geo	Bore hole on lateral surface	103
G311-Geo	Linear slot, lateral surface	104
G312-Geo	Circular groove on cylindrical surface	104
G313-Geo	Circular groove on cylindrical surface	104
G314-Geo	Full circle on cylindrical surface	105
G315-Geo	Rectangle on cylindrical surface	105
G317-Geo	Eccentric polygon on lateral surface	105
G411-Geo	Linear pattern on lateral surface	106
G412-Geo	Circular pattern on lateral surface	106

Overview of G commands in the MACHINING section

Tool positioning without machining		Page
G0	Positioning in rapid traverse	110
G14	Approach to tool change position.	110
G701	Rapid traverse to machine coordinates	111
Simple Linear and Circular Movements		Page
G1	Linear path	111
G2	Circular arc with incr. center dimensioning	112
G3	Circular arc with incr. center dimensioning	112
G12	Circular arc with abs. center dimensioning	112
G13	Circular arc with abs. center dimensioning	112
Feed Rate and Spindle Speed		Page
Gx26	Speed limit *	113
G48	Acceleration (slope)	113
G64	Interrupted feed	113
G192	Feed per minute for rotary axis	113
Gx93	Feed per tooth *	114
G94	Feed per minute	114
Gx95	Feed per revolution	114
Gx96	Constant cutting speed	114
Gx97	Spindle speed	114
Cutter radius compensation (TRC/MCRC)		Page
G40	Switch off TRC/MCRC	115
G41	TRC/MCRC left	115
G42	TRC/MCRC right	115
Zero point displacement		Page
Overview of datum shifts		116
G51	Datum shift (relative)	116
G53	Parameter-dependent datum shift	116
G54	Parameter-dependent datum shift	116
G55	Parameter-dependent datum shift	116
G56	Additive datum shift	117
G59	Absolute zero point shift	117
G121	Contour mirroring/shifting	117
G152	Zero point displacement, C axis	148
G920	Deactivate datum shifts	172

Zero point displacement		Page
G921	Activate datum shifts, tool lengths	172
G980	Activate datum shifts	172
G981	Activate datum shifts, tool lengths	172
Oversizes, safety clearances		Page
G47	Set safety clearances	118
G50	Switch off oversize	118
G52	Switch off oversize	119
G57	Paraxial oversize	119
G58	Contour-parallel oversize	119
G147	Safety clearance (milling)	119
Tools, types of compensation		Page
T	Tool change	120
G148	(Changing the) cutter compensation	120
G149	Additive correction	120
G150	Compensate right tool tip	121
G151	Compensate left tool tip	121
G710	Adding tool dimensions	121
SimpleTurning Cycles		Page
G80	End of cycle	134
G81	Simple longitudinal roughing	134
G82	Simple face roughing	135
G83	Contour repeat cycle	136
G85	Undercut	137
G86	Simple recessing cycle	138
G87	Transition radii	139
G88	Chamfers	139
Contour-based turning cycles		Page
G810	Longitudinal roughing cycle	122
G820	Face roughing cycle	124
G830	Contour-parallel roughing cycle	126
G835	Contour-parallel with neutral tool	127
G860	Universal recessing cycle	128
G866	Simple recessing cycle	129
G869	Recess turning cycle	130
G890	Finishing cycle	132

* "x" = number of spindles (0...3)

Thread cycles	Page
G31 Thread cycle	140
G32 Single thread cycle	141
G33 Single thread cut (Thread single path)	142

Drilling cycles	Page
G36 Tapping	146
G71 Simple drilling cycle	143
G72 Boring, countersinking, etc.	144
G73 Tapping cycle	145
G74 Deep-hole drilling cycle	147

C-axis machining

C axis	Page
G119 Select C axis (No. of C axis)	148
G120 Reference diameter, lateral surface machining	148
G152 Zero point displacement, C axis	148
G153 Standardize C axis	148

End face machining	Page
G100 Rapid traverse, face	149
G101 Linear path, face	149
G102 Circular arc, face	149
G103 Circular arc, face	149

Lateral-surface machining	Page
G110 Rapid traverse, lateral surface	150
G111 Linear path, lateral surface	151
G112 Circular arc, lateral surface	151
G113 Circular arc, lateral surface	151
G120 Reference diameter, lateral surface machining	148

Milling cycle group	Page
G840 Contour milling	152
G845 Pocket milling – roughing	156
G846 Pocket milling – finishing	157

Special functions

Assigning the Contour to the Operation	Page
G99 Workpiece group	110

Chucking equipment in simulation	Page
G65 Display chucking equipment	159

Slide synchronization	Page
G62 One-sided synchronization	160
G63 Synchronous start of slides	160
G162 Set synchronizing mark	160

Spindle synchronization, workpiece transfer	Page
G30 Converting and mirroring	169
G121 Contour mirroring/shifting	117
G720 Spindle synchronization	161
G905 Measuring C-angle offset	161
G906 Angular offset during spindle synchronization, measuring	161
G916 Traversing to a fixed stop	162
G917 Controlled parting using lag error monitoring	162
G991 Controlled parting using spindle monitoring	163
G992 Values for controlled parting	164

Contour regeneration	Page
G702 Storing/loading contour follow-up	164
G703 Contour follow-up ON/OFF	164
G706 K default branch	164

In-process/post-process measuring	Page
G910 Activate in-process measuring	165
G912 Actual-value determination for in-process measuring	165
G913 Deactivate in-process measuring	165
G914 Deactivate probe monitoring	165
G915 Post-process measuring	166

Load monitoring	Page
G995 Define monitoring zone	167
G996 Type of load monitoring	167

Other G functions		Page
G4	Period of dwell	168
G7	Precision stop ON	168
G8	Precision stop OFF	168
G9	Block precision stop	168
G15	Move rotary axes	168
G30	Converting and mirroring	169
G60	Deactivate protection zone	169
G98	Assignment spindle – workpiece	169
G121	Contour mirroring/shifting	117
G204	Waiting for moment	170
G717	Update nominal values	170
G718	Move lag error	170
G901	Actual values in variables	170
G902	Datum shift in variables	170
G903	Lag error in variables	170
G907	Block speed monitoring off	171
G908	Feed override 100%	171
G909	Interpreter stop	171
G918	Look ahead ON/OFF	171
G919	Spindle override 100%	171
G920	Deactivate datum shifts	172
G921	Activate datum shifts, tool lengths	172
G975	Lag error limit	172
G980	Activate datum shifts	172
G981	Activate datum shifts, tool lengths	172
Data input and data output		Page
INPUT	Input (# variables)	173
WINDOW	Open output window (# variables)	173
PRINT	Output (# variables)	173
INPUTA	Input (V variables)	174
WINDOWA	Open output window (V variables)	174
PRINTA	Output (V variables)	174

Programming variables		Page
# variables	Evaluation during program interpretation	175
V variables	Evaluation during program run	177
Program branches, program repeats		Page
IF.THEN..	Program branch	180
WHILE	Program repeat	180
SWITCH..CASE	Program branch	181
Special functions		Page
\$	Slide code	181
/	Skip level:	181
Subprograms		Page
Subprogram call		182
See Machine Manual		
G500..502	"OEM cycle"	
G600, 602..699	"PLC function"	
See Technical Manual		
G715	Real-time coupling function	
G716	Real-time coupling function	
G719	Real-time coupling function	
Reserved for internal use		
G16	Reserved for 3-D	
G704	Return inspection	
G705	Return inspection	
G900	Return inspection	
G990		

TURN PLUS soft keys (selection)

General soft keys

	Find input parameters by digitizing
	Calculate input parameters with pocket calculator
Increment	Incremental dimensions
	Switch to arc input
	Switch to line input
	"Continue" soft key – next element, next selection, etc.
	Tangential transition to next contour element
	Tangential transition to next contour element
Save contour	Save a contour

Element-selection soft keys

	Activate range selection
	Select next/previous contour element
	Select next/previous contour element
	Activate selection of multiple elements and select all elements
	Activate selection of multiple elements

Point-selection soft keys

	Activate multiple selection and select all elements
	Activate multiple selection
	Select next/previous point (contour transition)
	Select next/previous point (contour transition)

Soft keys: Center/end-point selection

	Center/end-point selection
	Next/previous center/end point
	Select next/previous center/end point

Soft keys: Form-element selection

	Select all form elements
	Select next/previous form element
	Select next/previous form element

Soft keys: General selection

	<input type="checkbox"/> Select marked element/point <input type="checkbox"/> Confirm selection
	Cancel selection of marked element/point